Autodesk Certified Professional in Simulation for Static Stress Analysis

Exam objectives

Target audience

The Autodesk Certified Professional (ACP) certification is designed for candidates who have advanced skills and can solve complex challenges in workflow and design. This type of experience typically comes from having worked with the software on a regular basis for at least two years, or equivalent to approximately 400 hours (minimum) to 1,200 hours (recommended) of Autodesk software experience. Certification at this level demonstrates a comprehensive skill set that provides an opportunity for individuals to stand out in a competitive job market.

Candidates who obtain this certification will have demonstrated advanced skills in simulation for linear static stress using Autodesk Fusion 360. They will have exhibited an understanding of preparing a model for simulation, configuring, solving, and interpreting results for a linear static stress simulation study. Earning this certification can help differentiate candidates in the job market by validating their skills in advanced computer-aided design (CAD) and computer-aided engineering (CAE) for linear static stress.

Prerequisite skills

It’s expected that candidates will already know how to:

- Navigate the user interface.
- Identify areas of the browser.
- Transition through various environments.
- Understand available file types.
- Display a part or assembly.
- Identify various planes and axes.
- Create fully constrained sketches.
- Create basic solid and surface features.
- Use the simplify workspace in a simulation to modify and remove features,

bodies, or components
- Create a Static Stress Simulation Study.
- Modify simulation study general and mesh settings.
- Manage and modify study materials.
- Define structural force, pressure, gravity, and moment loads.
- Define a structural fixed, pin, and frictionless constraint.
- Define a bolted connection.
- Solve automatic contacts.
- Manage component contacts.
- Modify local mesh settings.
- Run a simulation pre-check.
- Solve a simulation study.
- Interpret simulation results.
- Use simulation results tools.

Exam objectives

Here are some topics and software features that may be covered in the exam.

1. Simulation prep

1.1. Determine, evaluate, and interpret relevant information before model prep

1.1.a. Determine material appropriateness for linear static stress

1.1.b. Identify sufficient and appropriate loads

1.1.c. Classify load types

i. May include hydrostatic pressure, remote load, and bearing.

1.1.d. Interpret constraint information

1.1.e. Classify assembly connections

1.1.f. Evaluate sufficiency of information to complete analysis

1.1.g. Prioritize simulation information for analysis

i. May include structural vs. functional and material vs. appearance.

1.2. Apply simplify workspace tools to modify and remove features, bodies, or components

1.2.a. Use direct modeling tools to prepare a model for simulation
   i. May include split face, split body, and remove features and faces.
1.2.b. Clone and reuse simulation models
1.2.c. Confirm CAD models
1.2.d. Prioritize geometry features for analysis
1.2.e. Use inspection tools to validate component interference

1.3. Recall the requirements for importing appropriate CAD file types

1.3.a. Identify appropriate file types for running simulation studies
1.3.b. Capture design history

1.4. Select tools to repair an imported model

1.4.a. Clean up imported assembly structure
1.4.b. Isolate key components for an imported design
   i. May include Ground, Derive, and Remove.
1.4.c. Create manifold bodies from open surfaces

2. Simulation set up

2.1. Define and/or modify a static stress simulation study

2.1.a. Modify simulation study general and mesh settings
2.1.b. Determine when a linear static study is appropriate
2.1.c. Remove rigid body modes
2.1.d. Define appropriate element order
2.1.e. Select analysis type
   i. May include modal, stress, and displacement.

2.2. Assign and/or modify properties in Materials Library

2.2.a. Identify required mechanical information
2.2.b. Identify appropriate material properties
   i. May include a list of key material properties.
2.2.c. Customize generic material

2.3. Define local and global loads

2.3.a. Convert an analysis diagram into applied loads
2.3.b. Convert written description of load scenario into applied loads
2.3.c. Apply loads accurately to model

   i. May include bearing load, cyclic loads, pressure, normal and vector forces, gravity, and weight.

2.3.d. Select appropriate vector

2.3.e. Replace non-modeled bodies with remote forces

2.3.f. Differentiate structural constraints and loads

2.4. Define study and model constraints

2.4.a. Convert an analysis diagram into structural constraints

2.4.b. Convert a written description of structural constraints

2.4.c. Apply structural constraints accurately

   i. May include sliding contact, friction, bolted connection, fixed, pin, rigid body connector, universal, and directional.

2.4.d. Differentiate structural constraints and loads

2.4.e. Apply frictionless constraints to symmetry models

2.4.f. Explain limitations of structural constraints

2.5. Generate and manage contact sets

2.5.a. Generate contact sets between models

2.5.b. Assign appropriate contact tolerance

2.5.c. Understand appropriate contact type for application

2.5.d. Configure appropriate options for contact set

   i. May include select bodies, contact type, penetration type, and max activation distance.

3. Solve simulation

3.1. Review and address simulation pre-check information

3.1.a. Interpret common errors and warnings and predict their impact

3.1.b. Conclude whether common errors and warnings violate the constraints of a linear static analysis

3.1.c. Take appropriate action to address common errors and warnings

3.1.d. Generate and inspect mesh elements for a study

3.1.e. Ensure pre-check information aligns with analysis plan

4. Result Analysis

4.1. Obtain requested information using simulation results tools

4.1.a. Identify stresses at a specific feature or region in the model

i. May include slice plane, surface probe, and point probe.

4.1.b. Recognize and interpret expected data ranges and results when given a Results View

i. May include identifying maximum magnitude of stress, i.e., compression and tension, the difference between scale and plotted deflection, determining whether material is permanently deformed, and/or maximum vs. displayed deflection.

4.1.c. Determine appropriate stresses to display in Results View

4.2. Interpret results to evaluate the credibility of the simulation configuration

4.2.a. Evaluate credibility of results of setup

i. May include comparing results with expected action forces or an analysis diagram, identifying missing or misconfigured forces, constraints, and contact sets, identifying if forces are in the wrong direction or contact pressure, and evaluating the impact of geometry simplifications.

4.2.b. Evaluate credibility of result requirements

i. May include assessing whether results align with the limitations of linear static stress analysis, including small deformations, linear material behavior, and no permanent deformation.

4.2.c. Interpret model-induced stress concentrations

5. Study refinement

5.1. Determine method for local and/or global mesh setting refinements

5.1.a. Make a global change to the mesh

i. May include adjusting mesh density, creating curved mesh elements, and using mesh settings to improve transition with local mesh control, including max adjacent mesh size ratio, and/or configure adaptive mesh.

5.1.b. Use the local mesh control tools to refine a mesh around a feature or region of interest

5.2. Identify model geometry changes to improve subsequent iterations of the simulation

5.2.a. Identify model features that need to be suppressed, unsuppressed, or modified for analysis based on results

5.2.b. Select appropriate workflow(s) for modifications

5.3. Identify opportunities for the reuse and refinement of simulation studies

5.3.a. Identify the correct workflow for the reuse of simulation setup

i. May include clone load cases, simulation models, loads, and constraints.

5.3.b. Create a simulation based on a model simplification that retains simulation setup

i. May include clone simulation models.

5.3.c. Refine a simulation study with modified boundary conditions

i. May include clone load cases, loads, and constraints.

5.3.d. Create a simulation to accommodate a significant design change

i. May include clone simulation model or component from distributed design; reuse, redefine, or edit loads and constraints to new simulation model.