GENERAL

ansys workbench fatigue analysis tutorial

Published: September 19, 2026 | Index ID: #-

Introduction

When designing components that will endure repetitive loading, predicting and mitigating fatigue failure is paramount. Engineers rely on robust simulation tools to evaluate how materials behave under cyclic stresses, and **Ansys Workbench** has become one of the most popular platforms for this purpose. This comprehensive guide—an **ansys workbench fatigue analysis tutorial**—takes you through every step of the workflow, from setting up the model to interpreting the results. Whether you are new to finite element analysis (FEA) or looking to refine your fatigue simulation skills, this article will equip you with practical knowledge and actionable insights.

Understanding Fatigue Analysis in Ansys Workbench

What Is Fatigue?

Fatigue refers to the progressive, localized structural damage that occurs when a material is subjected to cyclic loading. Even stresses well below the ultimate tensile strength can initiate cracks, which grow over time until catastrophic failure. The key metrics in fatigue analysis are:

- Stress Range (9CÃ): The difference between the maximum and minimum stresses in a cycle.
- Mean Stress (<2 ,“ The average stress over a cycle, influencing crack growth rates.
- Number of Cycles (N): How many load cycles the component will experience during its life.

Why Use Ansys Workbench for Fatigue?

Ansys Workbench offers an integrated environment that streamlines the entire simulation process. Its fatigue module supports several material models—such as Miner’s Rule, the Palmgren–Miner cumulative damage model, and the Coffin–Manson equation—allowing users to choose the most appropriate approach for their application. Additionally, the graphical interface simplifies the setup of complex loading conditions, boundary constraints, and post processing visualizations.

Prerequisites for a Successful Tutorial

Software Requirements

- Ansys Workbench 2024 R1 or later (ensure the Fatigue module is installed).
- Ansys Mechanical for structural analysis.
- Optional: Ansys DesignModeler or SpaceClaim for geometry creation.

Hardware and System Specs

While small models can run on a standard laptop, large or highly refined meshes may require:

- Quad-core CPU with a minimum of 3 /GHz clock speed.
- 16 /GB RAM (32 /GB recommended for complex fatigue studies).
- Dedicated GPU for accelerated rendering (optional but beneficial for post-processing).

Basic Knowledge Needed

Familiarity with the following concepts will help you navigate the tutorial efficiently:

- Finite Element Method (FEM) fundamentals.
- Basic mechanical properties of materials.
- Understanding of cyclic loading and stress analysis.

Step by Step Fatigue Analysis Tutorial

1. Create a New Project

1. Launch Ansys Workbench.
2. Click File > New to start a fresh project.
3. Drag the Fatigue system from the toolbox onto the Project Schematic.

2. Import or Build Geometry

1. Double click the Geometry cell to open the geometry editor.
2. Use DesignModeler or SpaceClaim to create your component, or import an existing CAD file (STEP, IGES, Parasolid, etc.).
3. Ensure the geometry is clean: no duplicate surfaces, gaps, or overlapping elements.
4. Save and close the editor.

3. Define Materials and Properties

1. Open the Engineering Data database.
2. Search for your material (e.g., 7075 T6 aluminum) or create a new material entry.
3. Enter the following key properties:

Elastic modulus (E)

4. Poisson’s ratio (;Ò•
5. Yield strength (<1Ör•
6. Ultimate tensile strength (<1ÖB•
7. Fatigue strength coefficient (<2 &b•
8. Fatigue strength exponent (b)
9. Cyclic hardening/softening parameters (if using advanced models)

4. Set Up the Structural Analysis System

1. Drag a Static, General analysis system onto the schematic.
2. Link the Mechanical cell of the fatigue system to the Static, General cell by dragging a connector.
3. Double click the Static, General cell to open the solver.

5. Apply Loads and Boundary Conditions

- Define fixed or symmetry boundary conditions to simulate realistic supports.
- Apply cyclic loads:

Use the Load cell to add a Pressure, Force, or Moment.

- In the Load settings, set the Amplitude to a Step or Function that represents your loading cycle.
- Specify the Load Type as Static with Amplitude set to “Cyclic” to enable the fatigue module.

6. Mesh the Model

1. Open the Mesh cell.
2. Choose an element type suitable for fatigue (e.g., Solid tetrahedral or hexahedral elements).
3. Refine the mesh in critical regions (high stress gradients, stress concentrations).
4. Use Mesh Controls such as Size, Growth Rate, and Element Shape to balance accuracy and computational

cost.

5. Run the mesh and verify that the element quality metrics (aspect ratio, skewness) are within acceptable limits.

7. Configure Fatigue Study

1. Return to the Fatigue cell in the schematic.
2. Double click to open the fatigue module.
3. Under Fatigue Analysis Settings, choose the desired fatigue model:

Miner's Rule (cumulative damage).

4. Coffin–Manson (low cycle fatigue).
5. Paris Law for crack growth analysis.

8. Run the Simulation

1. Click the Compute button on the Fatigue cell.
2. Monitor the solver progress; typical fatigue runs may take longer due to the iterative nature of damage accumulation.
3. Once completed, review any warnings or errors and adjust settings as needed.

9. Post Processing Results

- Open the Results tree in the Mechanical interface.
- Explore the following key plots:

Stress Range Distribution (9CÃ).

- Damage Accumulation (D).
- Remaining Life (N_remaining).
- Fatigue Strength curves.

10. Validate and Iterate

Validate your simulation by comparing the predicted fatigue life with experimental data or design criteria. If discrepancies arise, consider:

- Refining the mesh.
- Adjusting material fatigue parameters.
- Re evaluating loading assumptions.
- Using a different fatigue model.

Tips and Best Practices

Mesh Quality for Fatigue

Fatigue analysis is highly sensitive to stress gradients. Ensure element sizes are small enough to capture sharp stress concentrations, especially near holes, notches, or fillets. A good rule of thumb is to have at least 10–15 elements across the critical dimension.

Choosing the Right Fatigue Model

Match the model to your application:

- Miner's Rule works well for high cycle fatigue where damage accumulates linearly.
- Coffin–Manson is preferred for low cycle fatigue with significant plasticity.
- Paris Law is ideal for crack growth studies when you have crack propagation data.

Using RCM and MTS

Residual Stress Compensation (RCM) and Multiaxial Stress Tensor (MTS) options can improve accuracy for components subjected to complex loading. Enable RCM to account for residual stresses from manufacturing processes.

Sensitivity Analysis

Perform a parametric study to assess how variations in material properties, loading amplitude, or geometry affect fatigue life. Use Ansys Workbench's **Parametric Study** feature to automate this process.

Common Pitfalls and How to Avoid Them

- Inadequate Mesh: A coarse mesh may overlook critical stress risers, leading to overestimated fatigue life.
- Wrong Material Data: Ensure fatigue parameters (<2 &`, b, c, m) are accurate for the specific material grade and heat treatment.
- Ignoring Mean Stress: Mean stress can significantly influence crack initiation; incorporate it using the Mean Stress Correction options.

Tags: #ansys #workbench #fatigue #analysis #tutorial