GENERAL

ansys workbench 14 static structural tutorials npgmbh

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

ANSYS Workbench 14 Static Structural Tutorials from npgmbh: A Complete Guide

When engineers and researchers need to predict how a structure behaves under static loads, ANSYS Workbench 14 offers a powerful, integrated environment to model, solve, and visualize results. This article provides an in depth walkthrough of the static structural workflow, enriched with tutorials curated by npgmbh. Whether you're a beginner setting up your first finite element model or an experienced user looking to refine your skills, this guide covers every step—from building the geometry to interpreting post processing data.

1. Understanding ANSYS Workbench 14

ANSYS Workbench 14 is a version of the ANSYS suite that emphasizes modularity and user friendly interfaces. It brings together several components—design exploration, physics modeling, and post processing—into a single workspace. The static structural module, one of the most frequently used modules, focuses on linear and nonlinear elasticity problems, buckling, and contact analysis.

Key features of Workbench 14 include:

- Drag and drop component placement.
- Integrated meshing with control over element type and size.
- Advanced solver options for both linear and nonlinear analyses.
- Post processing tools such as contour plots, displacement vectors, and stress overlays.

Because ANSYS Workbench 14 remains popular among universities and industry labs, many tutorial repositories, including those by npgmbh, still reference this version.

2. Why Static Structural Analysis Is Essential

Static structural analysis helps engineers answer questions like:

- Will a beam withstand a given load without excessive deflection?
- Where will the maximum stress occur in a bridge component?
- How does a change in material properties affect overall performance?

By solving equilibrium equations under static loads, designers can validate safety, reduce material usage, and optimize manufacturability. In the context of ANSYS Workbench 14, the static structural module streamlines this process, providing a clear path from geometry to results.

3. Setting Up Your First ANSYS Workbench 14 Project

3.1 Launching Workbench and Adding the Static Structural System

Open ANSYS Workbench 14 and create a new project. In the Toolbox panel, locate **Static Structural** under the "Analysis Systems" section and drag it into the Project Schematic. Double click the system to open the workflow tabs: Design Tree, Geometry, Model, Mesh, Solve, and Results.

3.2 Importing Geometry

You can import CAD files (STEP, IGES, Parasolid) or create geometry directly within DesignModeler. For beginners, a simple 2 D or 3 D shape—like a cantilever beam—provides an excellent starting point.

1. In the Geometry tab, click Import Geometry and select your file.
2. Verify the import by rotating the model and checking for errors.
3. Right click the geometry and choose Save to preserve the file in the project folder.

4. Defining Materials and Boundary Conditions

4.1 Assigning Materials

Static structural analysis requires accurate material properties. In the Model tab, right click **Engineering Data** and add a new material. Enter properties such as Young's modulus, Poisson's ratio, and density. For example, to model aluminum:

- Young's Modulus: 70 GPa
- Poisson's Ratio: 0.33
- Density: 2700 kg/m³

Assign the material to the entire geometry or specific components by selecting the faces or volumes and applying the material from the context menu.

4.2 Applying Loads and Constraints

In the Model tab, add **Loads** and **Constraints**:

1. Fixed Support – Select the surface or edge that will remain stationary.
2. Force – Apply a distributed or point load on the free surface.
3. Pressure – For fluid structure interaction scenarios.

Use the **Coordinate System** options to orient loads correctly. For a cantilever beam, fix one end and apply a downward force at the free end.

5. Meshing Strategies for Static Structural

5.1 Choosing Element Types

ANSYS Workbench 14 offers several element families:

- Solid Elements – 8 node linear (C3D8) or 20 node quadratic (C3D20).
- Shell Elements – 4 node (S4) or 9 node (S8).
- Beam Elements – 2 node (B31) or 4 node (B32).

For solid problems, start with linear tetrahedral elements (C3D4) and refine as needed. For thin structures, shell elements reduce computational cost.

5.2 Mesh Control Parameters

Right click the **Mesh** node and open the **Mesh Settings** dialog. Adjust:

- Element Size – Smaller elements capture stress gradients more accurately.
- Sizing Factor – Controls the global mesh density.
- Growth Rate – Limits how quickly element size changes across the mesh.

Use **Mesh Controls** to refine specific regions, such as load application points or stress concentrators.

6. Solving the Static Structural Model

6.1 Solver Options

Double click the **Solve** tab. In the Solver Settings panel, select the appropriate solver:

- Linear Solver – For small deformations and linear material behavior.
- Nonlinear Solver – If you anticipate large deformations, material nonlinearity, or contact.

Configure convergence criteria, such as displacement tolerance (e.g., 1e 6) and residual tolerance (e.g., 1e 5). These settings balance solution accuracy against computational effort.

6.2 Running the Analysis

Click **Run** to start the solver. Monitor progress in the **Solver Log** window. If the solver fails, review the error messages for missing boundary conditions or mesh irregularities.

7. Interpreting Post Processing Results

7.1 Visualizing Displacements

In the **Results** tab, open the **Deformed Shape** plot. Scale the deformation by a factor (e.g., 10) to make it visible. Use the **Vector Plot** to show displacement vectors and the **Contour Plot** to display magnitude.

7.2 Examining Stress Fields

Generate a **Contour Plot** for von Mises stress, maximum principal stress, or shear stress. Apply color ramps to highlight critical regions. Use **Cut Plane** views to inspect internal stress distribution.

7.3 Extracting Data

Right click the result and choose **Insert Table** to export data to Excel. You can also create **Probe Points** to monitor specific values during post processing.

8. Common Pitfalls and How to Avoid Them

- Inadequate Mesh – A coarse mesh can hide stress concentrations. Use mesh refinement near load points.
- Missing Boundary Conditions – Ensure all degrees of freedom are constrained; otherwise, the solver will issue rigid body modes.
- Material Property Errors – Double check units and values; a mis typed Young's modulus can drastically alter results.
- Solver Divergence – In nonlinear problems, start with a linear solve to get a good initial guess.

9. Advanced Topics in Static Structural Analysis

9.1 Nonlinear Material Behavior

For metals undergoing plastic deformation, define a Plasticity

Baca Juga (Artikel Terkait)

- **applied mechanics for engineering technology keith m walker**
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-keith-m-walker.pdf
- **applied mechanics for engineering technology answers**
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-answers.pdf
- **applied mechanics for engineering technology 8th edition solution manual**
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-8th-edition-solution-manual.pdf
- **applied mechanics for engineering technology 8th edition**
  URL: http://www.atemplatefree.com/applied-mechanics-for-engineering-technology-8th-edition.pdf

Tags: #ansys #workbench #static #structural #tutorials #npgmbh