GENERAL

ansys welding tutorial

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

ANSYS Welding Tutorial: Mastering Advanced Simulation for Real World Applications

Welding is a cornerstone of modern manufacturing, yet it remains one of the most complex processes to model accurately. Engineers need to predict heat distribution, residual stresses, distortion, and potential defect formation to design robust parts and optimize production. **ANSYS**provides a comprehensive, physics based platform that lets you simulate the entire welding cycle, from pre heat to post cooling. This tutorial walks you through every stage of an ANSYS welding simulation, offering practical tips, best practices, and real world examples to help you achieve reliable, repeatable results.

Why Simulate Welding with ANSYS?

Traditional trial and error methods can be costly and time consuming. By simulating the welding process:

- Reduce prototype count – Identify design flaws before building a physical part.
- Optimize process parameters – Find the sweet spot for heat input, travel speed, and shielding gas.
- Predict defects – Detect hot spots, cracking zones, and distortion early.
- Improve quality control – Generate data for statistical process control and lean manufacturing.

ANSYS integrates thermal, mechanical, and material behavior models, enabling a holistic view of welding performance. Whether you work with MIG, TIG, or laser welding, the simulation tools can adapt to your specific process.

Getting Started: The ANSYS Workbench Workflow

The ANSYS Workbench interface provides a structured environment where each component of the simulation lives in its own tab. A typical welding study includes:

1. Geometry – Build or import the part to be welded.
2. Material Library – Define thermal and mechanical properties for each material.
3. Mesh – Create a discretized representation of the geometry.
4. Welding Process – Set up heat source, path, and process parameters.
5. Boundary Conditions – Apply supports, thermal loads, and constraints.
6. Solution – Run the solver to compute temperature, stress, and displacement fields.
7. Post Processing – Visualize results, extract data, and generate reports.

Each step feeds into the next, so careful planning is essential. Below we dive into each stage in detail.

Step 1: Building or Importing the Geometry

High quality geometry is the foundation of any simulation. Use CAD tools such as SolidWorks, CATIA, or Creo to create a clean model. When importing, keep the following tips in mind:

- Remove unnecessary features – Fillets, chamfers, and small holes that do not affect the weld can be omitted to reduce mesh size.

- Use symmetry – If the weld geometry is symmetrical, model only a quarter or half of the part to save computational resources.
- Check for gaps and overlaps – These can cause mesh errors; use repair tools in ANSYS DesignModeler or SpaceClaim.

After the geometry is ready, import it into the **Geometry** cell of the Workbench. Validate the import by checking the model for missing surfaces or inverted normals.

Creating the Weld Path

The weld path is a critical input. It defines the trajectory of the heat source and influences the heat distribution. In ANSYS, you can define the path using:

- Pre defined lines – Create straight or curved lines that follow the desired weld seam.
- Point to point interpolation – Place points along the seam and let ANSYS generate a spline.
- Import from CAM – If you already have a G code or CNC program, import the path directly.

Ensure the path is smooth and continuous; discontinuities can lead to unrealistic heat spikes.

Step 2: Defining Materials and Thermal Properties

Welding simulations require accurate material models that capture temperature dependent behavior. ANSYS offers built in material libraries, but you can also create custom properties:

- Thermal conductivity – Varies with temperature; use piecewise functions or polynomial fits.
- Specific heat capacity – Important for transient thermal analysis.
- Density – Affects mass flow and heat transfer.
- Coefficient of thermal expansion – Critical for predicting distortion.
- Elastic modulus & Poisson's ratio – Needed for mechanical response.
- Yield strength & hardening curves – For plasticity and residual stress analysis.

When modeling a weld metal, you must also define the **fusion zone** and **heat affected zone (HAZ)** properties, which differ from the base material. ANSYS allows you to set these as separate material regions or use a **temperature dependent material model** that transitions automatically.

Material Assignment in ANSYS

After the material database is prepared, assign materials to each part or region:

1. Select the Model cell.
2. Drag the material from the Material Library onto the geometry.
3. Use Region or Material assignment to specify weld metal, base metal, and filler.

Verify the assignments by checking the **Material** tab in the model tree.

Step 3: Meshing – Balancing Accuracy and Efficiency

A good mesh captures gradients in temperature and stress while keeping the number of elements manageable. ANSYS offers several meshing strategies:

- Structured mesh – Good for simple geometries; provides high accuracy.
- Unstructured mesh – Flexible for complex shapes; uses tetrahedrons or polyhedra.
- Hybrid mesh – Combines structured and unstructured elements; ideal for weld zones.

Key meshing considerations:

- Refinement near the weld – Use size controls or growth rates to create a finer mesh in the fusion zone.
- Element type – Hexahedral elements are preferred for thermal analysis; tetrahedral elements work well for complex geometries.
- Aspect ratio – Keep it below 5 to avoid numerical inaccuracies.
- Mesh convergence – Perform a mesh study to ensure results are independent of element size.

In the **Mesh** cell, use the **Element Size** slider to control global refinement, then apply **Local Mesh Controls**to the weld zone for higher resolution.

Automatic vs. Manual Mesh Controls

ANSYS's **Mesh Control**tools can automatically detect high gradient regions based on temperature or stress. However, manual control often yields better results because you can target the exact weld area:

- Right click the weld region and select Insert Mesh Control.
- Choose Size or Growth Rate and set appropriate values.
- Apply Boundary Layer elements if you need to capture steep temperature gradients near the surface.

Step 4: Setting Up the Welding Process

ANSYS offers several heat source models that represent different welding techniques:

- Double Ellipsoid – Commonly used for MIG and TIG; captures both the front and rear heat distribution.
- Gaussian – Suitable for laser welding; provides a bell shaped heat profile.
- Point Heat Source – Simplified model for quick studies.
- Custom Heat Source – Define your own distribution based on experimental data.

To configure the heat source:

1. Drag the Weld system into the Model cell.
2. Double click the Weld system to open its settings.
3. Select the appropriate Heat Source Type.
4. Define the Heat Input (W) and Travel Speed (mm/s) based on the welding procedure.
5. Set the Weld Path by linking it to the geometry path you created earlier.
6. Optionally, add Pre Heat or Cooling steps to mimic real world conditions.

ANSYS also allows you to model **shielding gas convection** and **convective heat loss**from the weld pool, which can improve accuracy for high speed processes.

Time Dependent Simulation Settings

Welding is inherently transient. Configure the solver to capture the dynamic temperature field:

- Time Step – Choose a step size that captures the peak temperature but does not exceed solver stability limits.
- Simulation Duration – Set the total time to cover the entire weld cycle, including cooling.
- Solution Controls – Enable Adaptive Time Stepping to let ANSYS adjust steps during rapid temperature changes.

For large parts or long welds, consider splitting the simulation into multiple steps (e.g., **Welding**, **Cooling**) to reduce computational effort.

Step 5: Applying Boundary Conditions and Constraints

Boundary conditions dictate how the part interacts with its environment and supports. Typical constraints in welding simulations include:

- Fixed or Roller Supports – Prevent rigid body motion.
- Thermal Contacts – Model heat transfer between parts.
- Heat Loss – Convective and radiative cooling from surfaces.
- Pre Heat – Uniform temperature increase before welding.
- Symmetry Planes – Reduce model size if applicable.

To apply a constraint:

1. Select the Model cell.
2. In the Connections tab, choose the type of constraint.
3. Click on the part or surface where the constraint applies.
4. Set the values (e.g., fixed displacement, convection coefficient).

When modeling **thermal contact**, use the **Contact** interface to define thermal resistance or perfect contact. For **convective heat loss**, set the convection coefficient based on the surrounding fluid (air or gas) and the surface area.

Defining Support Conditions for Distortion Analysis

Distortion predictions require accurate support modeling. If you have a real fixture, replicate it in the model: