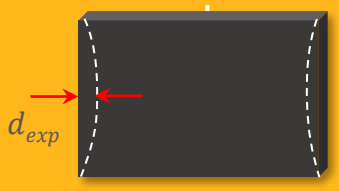
Ansys Additive Distortion Calibration Quick Start Guide

Use this Quick Start Guide to calibrate Ansys Additive software to match your machine/material scenario. The goal is to determine a calibration factor, called a Strain Scaling Factor (SSF), that compensates for the difference between a measured distortion and a simulated distortion. This guide describes the process for performing an Assumed Strain simulation type using either Linear Elastic or J2 Plasticity stress mode.

Step 1

Build & Measure

- Choose a calibration part that is easy to build and yields high distortion
- Build the calibration part with the same process parameters you plan to use for your part
- If possible, build the part directly on the baseplate to minimize support structures
- Allow enough room to make measurements while the part is still attached to the baseplate
- After fabrication, measure displacement (d_{exp}) at location of interest



Step 2

Simulate

- Run Assumed Strain simulation type with the same geometry and material
  - Import your calibration geometry
  - Choose your material
  - Set stress mode = Linear Elastic or J2 Plasticity
  - Set Strain Scaling Factor (SSF) = 1 (default)
  - Use default output options
  - Start the simulation
  - Export On plate stress/displacement
  - Obtain displacement (d_{sim}) at same points and same directional component (X or Y) as measured
- Run new simulation as above but with SSF_{new} calculated in Step 3

Step 3

Calculate & Compare

- Calculate new SSF:

$$SSF_{new} = \frac{d_{exp}}{d_{sim}} SSF_o$$

where SSF_o is previous SSF

- Compare difference between d_{sim} and d_{exp} until it converges toward zero or an acceptable level
- Record the final SSF_{new} value as $SSF_{cal\ LE}$ or $SSF_{cal\ J2}$
- Create custom materials with SSF_{cal} values in Additive Print

Learn more about Ansys Additive calibration at:
http://storage.ansys.com/doclinks/ansys.html?code=AddCalibration-ALU-M2a

Ansys