AD FALCON API Manual

Validation of FEM Code with a Three-Dimensional Foundation Model Using a Linear Elastic Material

Javad Ghorbani

March 26, 2026

Contents

ARTEMIS DEV

1 Validation of FEM Code with a Three-Dimensional Foundation Model Using a Linear Elastic Material

1.1 File Name

fem_3_3d_boussinesq.txt

1.2 Problem Description

This validation mirrors the loading and material assumptions used in the plane-strain and axisymmetric Boussinesq benchmarks, but runs the same elastic foundation problem in a full **three-dimensional** setting.

The objective is to validate the vertical stress distribution beneath the center of the loaded region by comparing:

- **3D FEM** results (vertical stress component `TotalStressZZ`)
- **Axisymmetric FEM** results (vertical stress component `TotalStressYY`)
- The **analytical Boussinesq** solution beneath the center of a uniformly loaded circular area

1.3 FEM Model Setup

- **Analysis type:** `ThreeDNonCoupled`
- **Element type:** `T10P4` (10-node tetrahedra)
- **Material:** linear elastic ($E = 210000$ kPa, $\nu = 0.3$)
- **Domain:** truncated elastic half-space represented by a $2 \times 2 \times 2$ m box
- **Mesh:** structured T10 tetrahedra (current deck: `nx=24, ny=24, nz=10` → `34560` elements)
- **Boundary conditions (rollers):**
  - Base: `DisZ = 0` at $z = 0$
  - Lateral faces: `DisX = 0` at $x = \pm 1$, `DisY = 0` at $y = \pm 1$
- **Load:** compressive surface pressure applied on the top surface ($z = 2$) over the central region (implemented using `@SurfacePressure` on T10 faces)
- **Step:** static ramp with a **fixed time increment** (`@@NumberSteps: 1`, `@@ModernAutoInc: No`)

1.4 Postprocessing (CSV extraction)

The input deck writes a CSV file using `% LineStateOutput`:

- Output file: `line_totalstresszz_3d.csv`
- Line: from `(0, 0, 1.95)` to `(0, 0, 0)` with spacing `0.02` (starts **5 cm below** the surface to avoid near-surface artifacts)
- Variable: `TotalStressZZ`

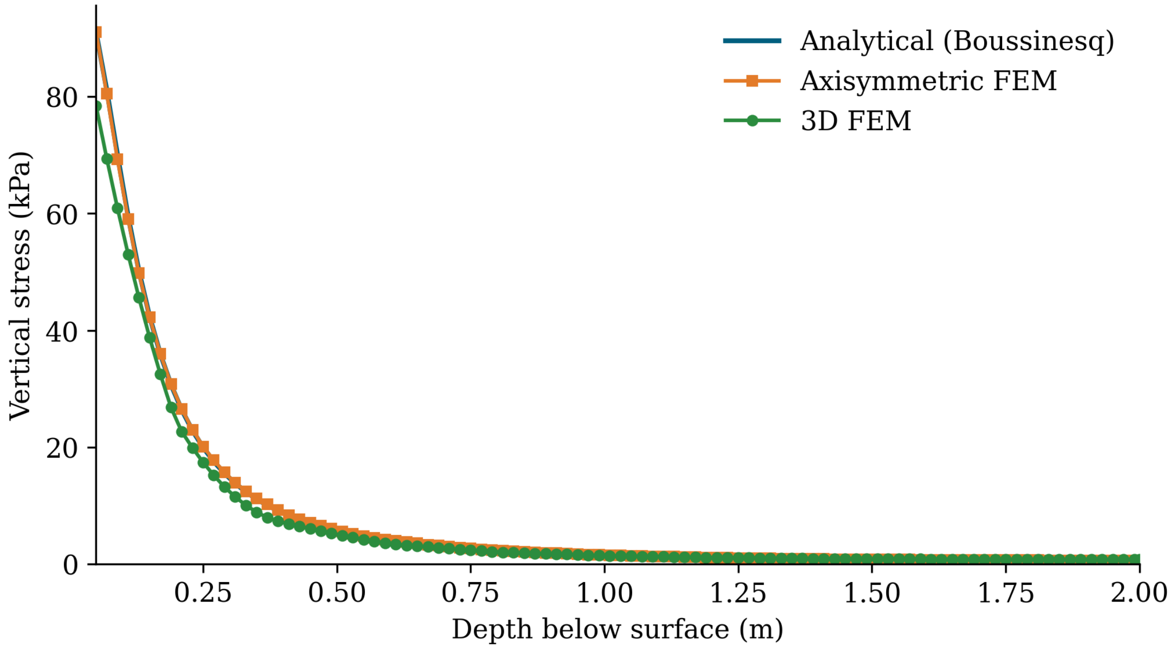

Figure 1: Figure: 3D vs axisymmetric vertical stress beneath the center.

The input deck also writes a center settlement time history using `% DOFOutput`:

- Output file: `dof_center_disz_3d.csv`
- DOF: `DisZ` at the node closest to the **resultant load center** on the top surface (computed from the coordinates of all nodes involved in the applied surface pressure; the node ID is written in the CSV header; for the current mesh it is exactly `(0,0,2)`).

1.5 Center settlement vs. axisymmetric

Using the same load/material parameters and reading the final-step center displacement:

- **3D:** `DisZ(0,0,2) = -7.4756e-05` m (`-0.07476` mm)
- **Axisymmetric:** `DisY(r=0,z=2) = -8.3538e-05` m (`-0.08354` mm)

So the 3D model predicts about **10.51% smaller** center settlement than the axisymmetric model for this finite, roller-bounded domain.

Much closer agreement is often achievable by further refining the 3D mesh (especially near the loaded area). Here the intent is primarily to demonstrate the 3D setup and postprocessing workflow; pushing the agreement further is left as a follow-up exercise.

1.6 Results

The plot below compares the vertical stress beneath the centerline (starting at **0.05 m depth**) against the axisymmetric benchmark and the analytical solution.