AD FALCON API Manual

Validation of FEM Code with an Axisymmetric Foundation Model Using a Linear Elastic Material

Javad Ghorbani

March 26, 2026

Contents

1 Validation of FEM Code with an Axisymmetric Foundation Model Using a Linear Elastic Material

1.1 File Name

fem_2_axisymmetric.txt

1.2 Problem Description

This example builds on **Example 1** but analyzes the foundation model under **axisymmetric conditions**. The geometry, material properties, and loading conditions remain the same as described in **Example 1**. Static mode is used alongside an autoincrement option.

The objective is to validate the vertical stress (σyy) distribution predicted by the FEM code against the analytical solution derived from the Boussinesq equations, using axisymmetric mechanics.

Refer to **Example 1** for the geometry and boundary conditions.

1.3 FEM Model Setup

- **Analysis Type:** Axisymmetric Non-coupled (AXUnCoupled). Element Type, Material Properties, Step Definition and Load Configuration are the same as **Example 1**

1.4 Results

The FEM solution for the vertical stress (σyy) distribution under axisymmetric conditions is compared to the analytical solution along the depth beneath the loaded area.

1.4.1 Comparison with Analytical Solution

The figure below shows the plot of σyy versus depth:

Figure 1. Predictions against analytical solution beneath the center of the footing under axisymmetric conditions.

1.5 Analytical Settlement at the Center (Flexible Circular Load)

For a uniformly loaded circular area on a linear-elastic, homogeneous, isotropic half-space (flexible load), the maximum surface settlement at the center is

$$w_{\max}(r = 0) = \frac{2\,(1 - \nu^2)}{E}\, q\, a$$

where q is the uniform pressure, a is the load radius, E is Young's modulus, and ν is Poisson's ratio.

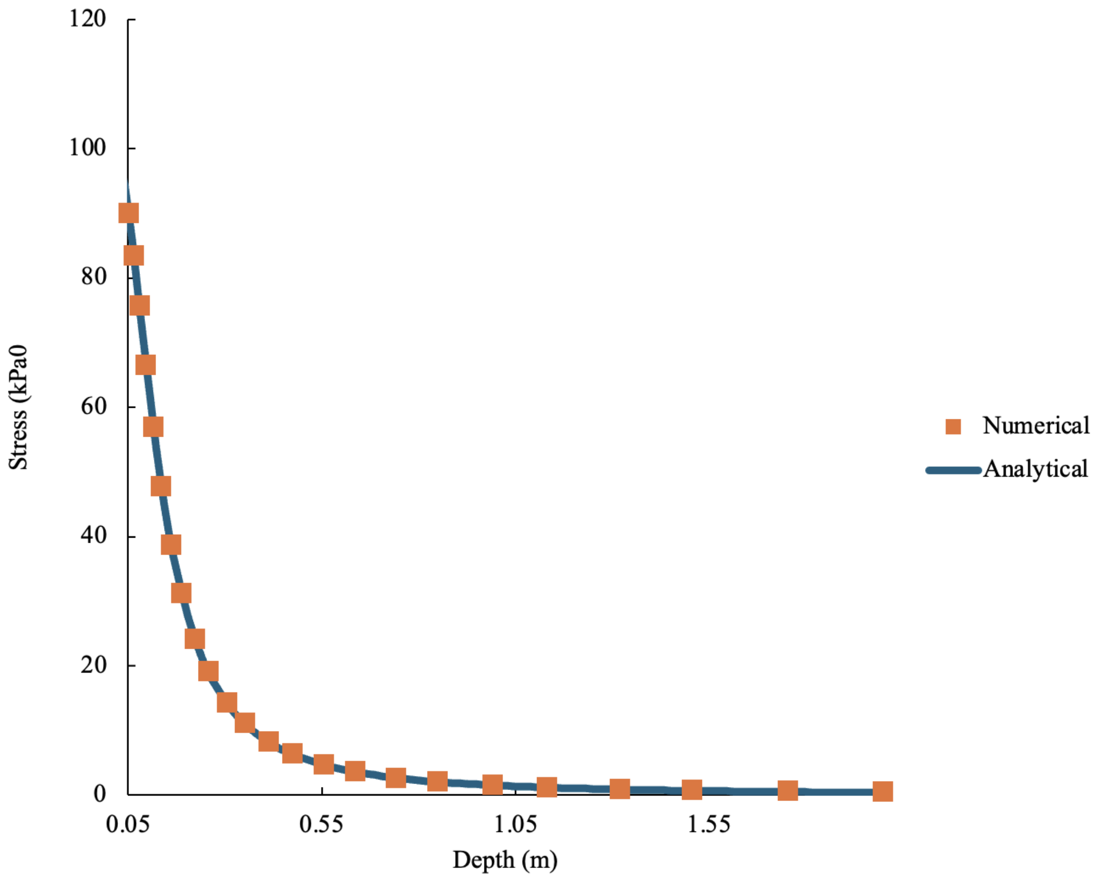

Figure 1: Figure 1: Predictions against analytical solution beneath the center of the footing.

Example with the properties used here: $E = 210{,}000$ kPa, $\nu = 0.3 \Rightarrow (1-\nu^2) = 0.91$, $q = 100$ kPa, $a = 0.10$ m:

$$w_{\max} = \frac{2 \times 0.91}{210000} \times 100 \times 0.10 = 8.6667 \times 10^{-5}\ \text{m} = 0.08667\ \text{mm}$$

1.5.1 Why the FEM settlement is slightly smaller

- Finite domain and rigid boundaries (box 2×2 m with $u_y = 0$ at the base) limit deep strains compared to an infinite half-space.
- Lateral restraints ($u_x = 0$ at vertical sides) restrict spreading beneath the load, increasing apparent stiffness.
- Minor discretization and element-formulation effects.

Comparison at the center: analytical $w_{\max} = 0.08667$ mm vs FEM $w_{\max} = 0.0835$ mm → about -3.65 % (FEM smaller for the reasons above).

In the provided input file (`fem_2_axisymmetric.txt`), the FEM settlement at the center is recorded via `% DOFOutput` to `dof_center_disy_axisymmetric.csv`.