AD FALCON API Manual

Analysis of Plane Strain Foundation Using a Linear Elastic Material

Javad Ghorbani

March 26, 2026

Contents

1 Analysis of Plane Strain Foundation Using a Linear Elastic Material

1.1 Problem Description

This validation case involves a 2×2 m foundation model analyzed using Plane Strain Uncoupled (PLUnCoupled) mechanics. A uniform pressure of 100 kPa is applied over a central 0.1 m width. Due to the axisymmetric nature of the problem, only half of the domain is modeled to improve computational efficiency.

1.1.1 Static Analysis in FALCON

Static analyses in FALCON can be conducted using two approaches:
- Option 1: Using a dynamic implicit solver with large time steps to suppress velocity and acceleration effects.
- Option 2: Using a static implicit solver, where time steps are pseudo-time increments rather than real-time evolution.

> Performance note: The dynamic option assembles and solves using the full set of matrices (including the mass and damping matrices in addition to the stiffness), which increases computational work per step. In practice, we observe the dynamic route to be approximately **2.6×** slower than the static route for the same mesh and step configuration. For genuinely quasi-static problems, prefer the static solver (or set `@@SimMode: Static`) to minimize runtime.

1.2 File Names

- Option 1 (Dynamic Implicit Solver with a Sufficiently Large Time Step)
- Option 2 (Static Implicit Solver, Fixed Time Increment)
- Option 3 (Static - Automatic Increment Solver)

1.2.1 Simulation Setup

- Total Analysis Time: 100,000 s
- Time Step Size: 1,000 s (100 fixed time steps)

The purpose of this validation is to compare the vertical stress σ_{yy} distribution predicted by the FEM code against the analytical solution derived from Boussinesq equations for a uniform load on a semi-infinite elastic medium.

1.2.2 Model Geometry

The figure below illustrates the foundation model geometry:

Figure 1: 2×2 m rectangular domain with a uniform pressure applied over a 0.1 m width.

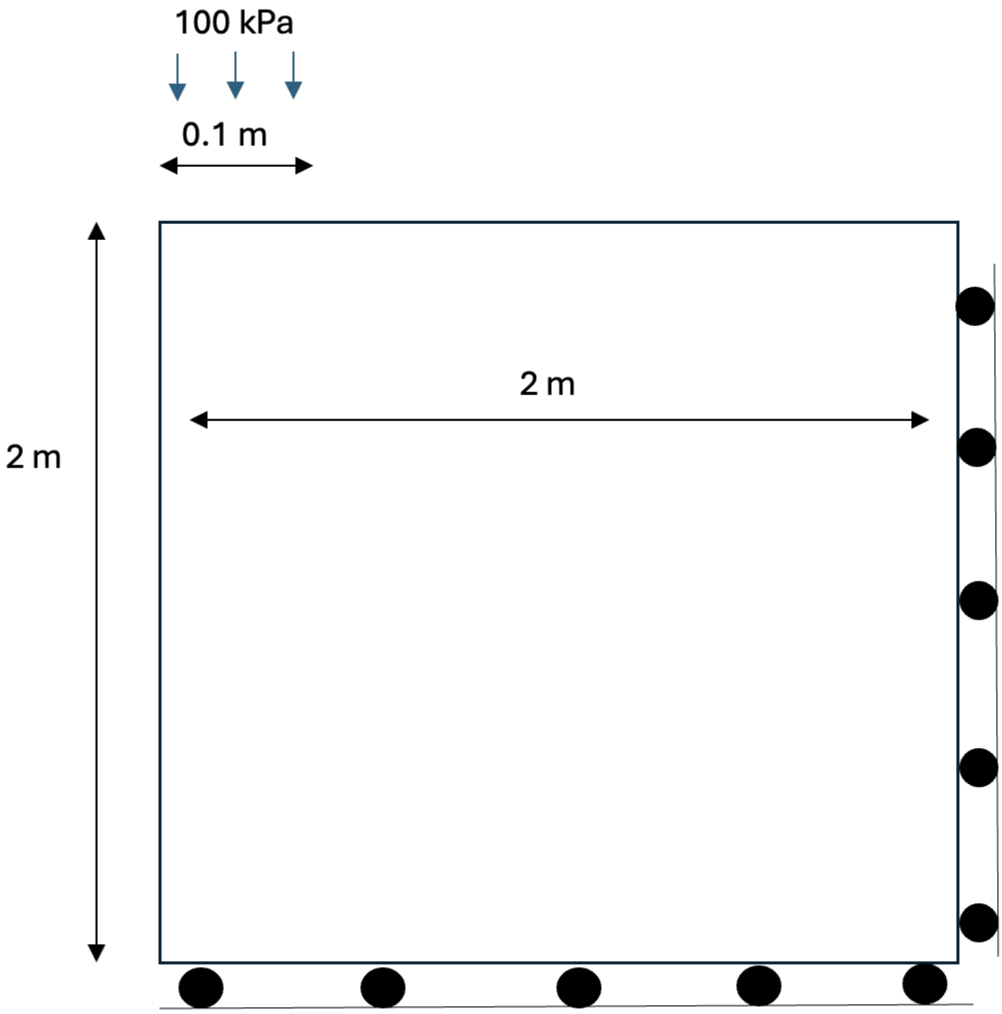

Figure 1: Figure 1: Geometry of the footing.

1.3 FEM Model Setup

- Analysis Type: PLNonCoupled (Plane Strain Non-Coupled Mechanics)
- Element Type: N6P6 (6-node elements with 6 integration points)

1.3.1 Geometry and Boundary Conditions

- Domain Size: 2 m × 2 m
- Boundary Conditions:
  - Lateral Boundaries: Restrained against horizontal movement $u_x = 0$
  - Bottom Boundary: Restrained against vertical movement $u_y = 0$
  - Symmetry Condition: Applied along x = 0
  - Uniform Pressure: 100 kPa over a 0.1 m width on the top surface

1.4 Material Properties

The foundation soil is modeled as a homogeneous, isotropic, linear elastic material with the following properties:

- Material: `Mat1`
- Young's Modulus E: 210,000 kPa
- Poisson's Ratio ν: 0.3

1.4.1 Mechanical Model Configuration

```
@Mech: Elastic YoungsModulus 210000 PoissonRatio 0.3
```

1.5 Step Definition

This analysis consists of one step where a ramp load is applied. The step definition in FALCON provides various control options for numerical simulations:

```
% Step Definitions
@Step 1:
@@SolverType: Direct
@@StartStep: 0
@@StepTime: 100000
@@NumberSteps: 100
@@OutputInterval: 10
@@OutputTypes: Displacement TotalStress
@@Geostatic: No
@@ErrorTarget: 1.e-3
%%%
```

1.5.1 Step Configuration Details

- **@@SolverType: Direct** → Specifies that a direct solver is used for numerical computations.
- **@@StartStep: 0** → Indicates that the analysis begins at step 0, ensuring proper initialization.
- **@@StepTime: 100000** → Defines the total duration of the analysis in seconds, ensuring quasi-static conditions.
- **@@NumberSteps: 100** → Breaks the total step time into 100 sub-steps for better numerical stability.
- **@@OutputInterval: 10** → Controls how frequently results are written to the output file (every 10 steps in this case).
- **@@OutputTypes: Displacement TotalStress** → Limits output data to displacement and total stress.
- **@@Geostatic: No** → Disables geostatic stress initialization since gravitational equilibrium is not considered in this setup.

1.5.2 Controlling Output Frequency (@@OutputInterval vs. @@OutputControlType)

FALCON supports two equivalent ways to control how often results are written:

1) Legacy (step-based)

```
@@OutputInterval: 10   # write results every 10 steps
```

- Parsed as step-based output. Must be an integer > 0.
- If invalid, the reader falls back to writing every step.

2) Modern (explicit control type + value)

```
@@OutputControlType:  ByStep
@@OutputControlValue: 10       # same effect as @@OutputInterval: 10
```

or time-based writing (by simulation time):

```
@@OutputControlType:  ByTime
@@OutputControlValue: 5000.0   # write every 5000 seconds of cumulative
simulation time
```

Notes - ByStep: write when (currentStep − startStep) is a multiple of the control value. - ByTime: write when the accumulated time since the step start increases by at least the control value. - Both control values must be > 0; defaults to step-based every step on invalid input. - Defaults (if unspecified): ByStep with interval = 1.

Examples for this case (StepTime = 100000, NumberSteps = 100 → each step = 1000 s) - ByStep, 25 → output at steps: 25, 50, 75, 100 - ByTime, 5000 → output approximately every 5 steps (since 5 × 1000 s = 5000 s)

Minimal step blocks illustrating both styles

```
% Step Definitions
@Step 1:
@@StartStep:        0
@@StepTime:         100000
@@NumberSteps:      100
@@SolverType:       Direct
@@OutputInterval:   25          # legacy, every 25 steps
@@OutputTypes:      Displacement TotalStress
%%%
```

```
% Step Definitions
@Step 1:
@@StartStep:           0
@@StepTime:            100000
@@NumberSteps:         100
@@SolverType:          Direct
@@OutputControlType:   ByTime
@@OutputControlValue:  5000.0     # write every 5000 seconds
@@OutputTypes:         Displacement TotalStress
%%%
```

Tip: For static analyses (pseudo-time), ByStep is often clearer. For consolidation or dynamic cases in real time, ByTime can be more intuitive.

Internals: The reader maps `@@OutputInterval:` to a step-based control. Explicit `@@OutputControlType:`/`@@OutputControlValue:` let you switch between step- and time-based controls.

Validation: Both inputs must be positive. If not, the parser silently defaults to ByStep with value 1.

Performance: Very frequent outputs can slow runs significantly due to I/O. Use conservative intervals during calibration, and refine only when needed.

Compatibility: Existing inputs using `@@OutputInterval:` remain supported and need no changes.

Post-processing: Output cadence determines the temporal resolution available in plots and animations.

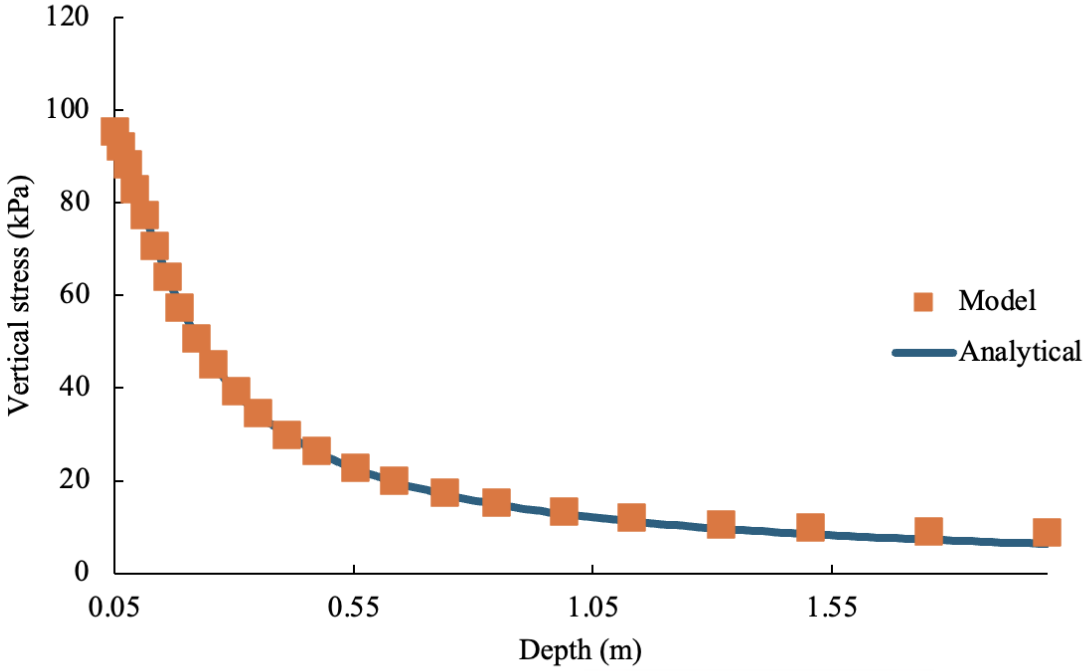

Figure 2: Figure 2: FEM vs. Analytical Solution.

See also: Steps and Simulation Modes.

Advanced: You can vary output control per step to capture transients in one step (fine-grained) while coarsening elsewhere.

Note: If **@@SimMode** is not explicitly given, the solver default simulation model is **Dynamic**.

1.6 Results

The FEM solution for the vertical stress σ_{yy} distribution is compared with the analytical solution using Boussinesq equations. The comparison is made along a vertical line passing through the center of the loaded area (x = 0).

1.6.1 Comparison with Analytical Solution

Figure 2: Comparison of FEM predictions and analytical solution beneath the center of the footing.

1.6.2 Option 2: Static Implicit Solver (Pseudo-Time Increments)

In this mode, time does not represent real seconds but serves as a substepping controller. The solver steps through pseudo-time to achieve a static equilibrium.

```
% Step Definitions
@Step 1:
@@StartStep:        0
@@StepTime:         1.0        # Total pseudo-time units
@@NumberSteps:      100        # Fixed pseudo-time increments
@@SolverType:       Direct
@@SimMode:          Static    # Enforces static condition in this step
@@OutputInterval:   10
@@OutputTypes:      Displacement TotalStress
@@ErrorTarget:      1.e-3
%%%
```

1.7 Option 3: Automatic Increment Solver

This option lets FALCON adjust the time-step increment automatically while solving the exact same plane-strain foundation model. For further explainations of variables used refer to Automatic Increment and Step Definition

```
% Step Definitions
@Step 1:
@@StartStep:             0
@@StepTime:              1.0
@@ModernAutoInc:         Yes
@@SolverType:            Direct
@@MaxIterations:         100
@@InitialStepIncrement:  1.0e-2
@@MinTimeStep:           1.0e-7
@@MaxTimeStep:           1.0e0
@@UseModifiedNewton:     No
@@UL:                    No
@@SimMode:               Static
@@OutputInterval:        1
@@OutputTypes:           Displacement TotalStress
@@ErrorTarget:           1.0e-3
%%%
```