Modelling of Moving Load on Embankment

For LUSAS version:	21.0
For software product(s):	LUSAS Bridge plus
With product option(s):	Geotechnical, Nonlinear

Problem Description

This example aims to study the impact of a load in motion on an embankment. The model includes four layers: soft rock, base and sub-base, and reinforced road bed. The total length of the load track being analysed is 50 meters.

Keywords

Moving Load, Settlement.

Associated Files

Associated files can be downloaded from the user area of the LUSAS website.

❑ **moving_load.lvb** carries out automated modelling of the example.

- Use **File > New** to create a new model of a suitable name in a chosen location.

- Use **File > Script > Run Script** to open the lvb file named above that was downloaded and placed in a folder of your choosing.

Objectives

The goal of this example is to demonstrate the process of incorporating a moving load into a model and determining the settlement of the load along its path.

Preparing the Model Features

The user has to create a new model, set the Analysis category as 3D, and specify the model units as kN,m,t,s,C.

Feature Geometry

The model is generated by creating surfaces on the XZ plane and then extrude them along the Y axis. By utilizing the Copy and Sweep command, the time required to create the model is considerably reduced. However, it is essential to be diligent in ensuring that the surfaces are properly connected and that there are no overlaps (Figure 1 and 2).
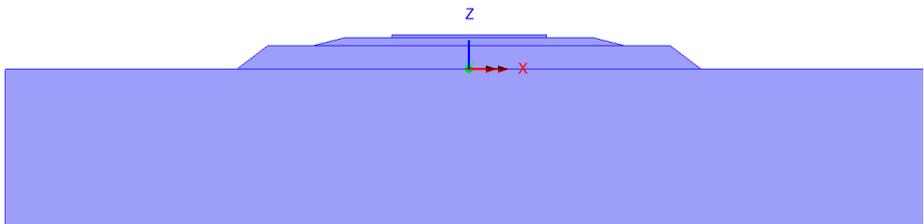
Figure 1: The model on XZ Plane
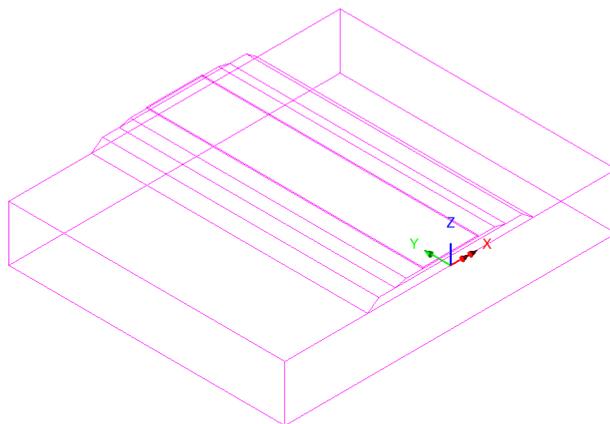
Figure 2: 3D Model

Preparing the Model Attributes

Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model. Figure 3 shows the attributes of model.
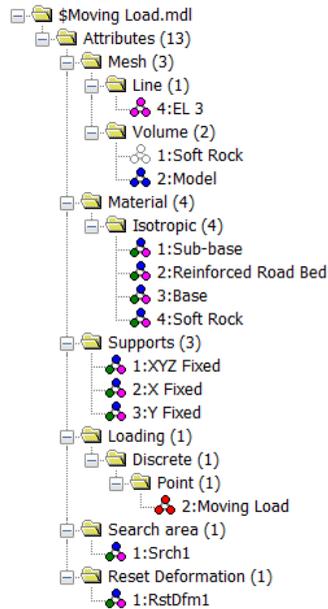
Figure 3: Model Attributes

Defining the Mesh

Hexahedral, quadratic stress elements (HX20) have been used for meshing (Figure 4)
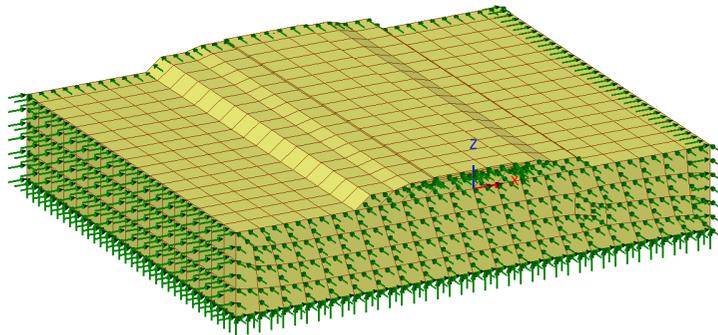
Figure 4: Mesh and boundary conditions

Defining the Materials

An isotropic elastic material is used for the reinforced road bed while other materials have been assigned nonlinear behaviour based on the Modified Mohr-Coulomb failure surface. All material properties are listed in table 1.

Table 1: material properties

Layer	Mass Density	Young's modulus, E	Poisson's ratio, ν	Angle of friction, ϕ	Angle of dilation, ψ	Cohesion, c	K_0
Road bed	1.9 t/m^3	2.1E6 kPa	0.25	-	-	-	0.333
Base	1.9 t/m^3	1.0E3 kPa	0.35	31°	0°	20 kPa	0.485
Sub-base	2.0 t/m^3	100.0E3kPa	0.3	40°	0°	1 kPa	0.357
Soft rock	2.2 t/m^3	1.2E6 kPa	0.28	37°	0°	10 kPa	0.398

Defining the Supports

The model is restrained in X and Y direction, and fully fixed supports have assigned to the base (Figure 4)

Defining the Loads

In addition to gravity, we also created point loads which apply concentrated force at specific points. These point loads can be defined individually or as a set of discrete point loads in 3D space, each with a different magnitude. In this example, all point loads are defined using a grid and have a value of 150 kPa (Figure 5).
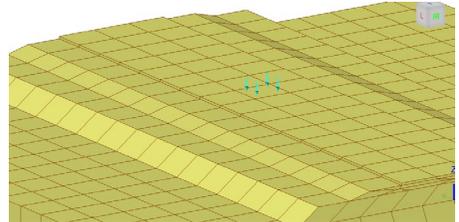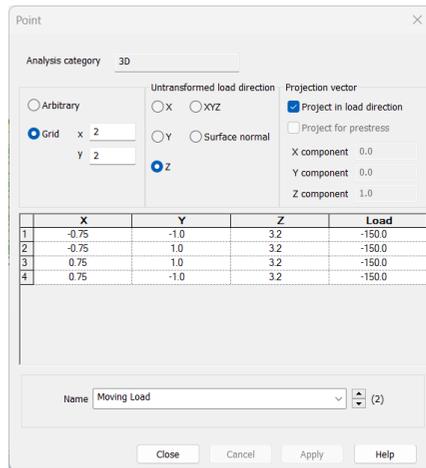
Figure 5: Discrete Points Load

Defining Other Attributes

When using a discrete loading to represent a vehicle passing across a structure it is necessary to target the application of the load onto the model using a **Search Area** (Figure 6).
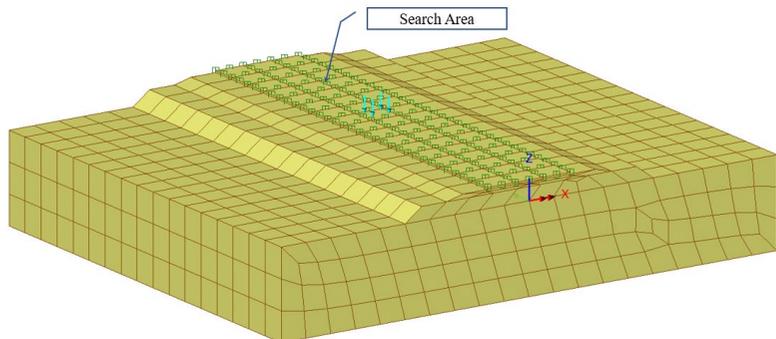
Figure 6: Discrete Points Load

Note. The distribution of load to the nodes follows the shape functions of the particular element. In quadratic elements, this distribution can appear at first unlikely. For example, a unit positive load at the centre of an 8-noded quadratic element, results in negative 0.25 loads at the corners and positive 0.5 loads at the mid-side nodes (LUSAS Help).

We reset the deformation in the first load case via **Reset Deformation** attribute.

Running the Analysis

We consider the following stages:

Initial Phase

This stage simulates the initial conditions where **Gravity** is applied as a load, the moving load has not yet initiated. Nonlinear analysis control properties are defined for this phase, all the parameters are left at their default values.

Moving Load Phases

In LUSAS, the moving load generator is used to create many load cases resulting from a vehicle moving along a defined path. To access this tool, use **Bridge > Moving Load Analysis**, which will bring up a dialog box with parameters such as "search area", "reference path," and "path division" (Figure 7). These parameters can be used to customise the load cases with the moving load. The search area "Srch1" has already been created (Figure 6) and the user can preselect the path of the moving load (line 93) before launching the dialog. There are 26 stages that represent the movement of the load along the path, which is illustrated in figure 8.

Figure 7: Moving Load Analysis
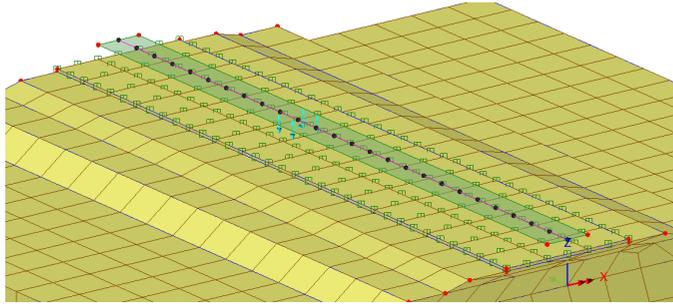
Figure 8: Moving load path

Viewing the Analysis

Analysis loadcase results are present in the Treeview.

Deformed Mesh

The deformed mesh at stage 11 is depicted in Figure 9.
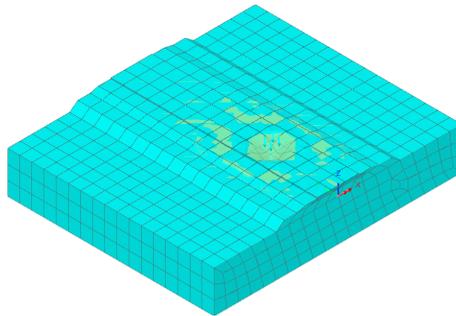
Figure 9: Deformed mesh at stage 11

Settlement

Since soil layers are homogenous, it is expected that the displacement along the moving load's path will be consistent, with a value of about 8-9mm as shown in Figure 10.
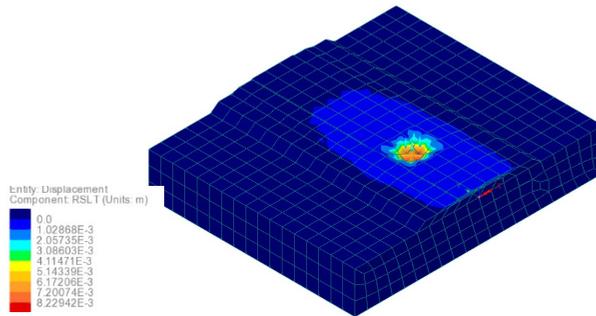
Figure 10: Resultant displacement at stage 11 (m)

Envelope

An overview of the maximum displacement along the tracking path can be obtained by enveloping loads along that path. This is done by selecting **Analysis > Envelope** (Figure 11).
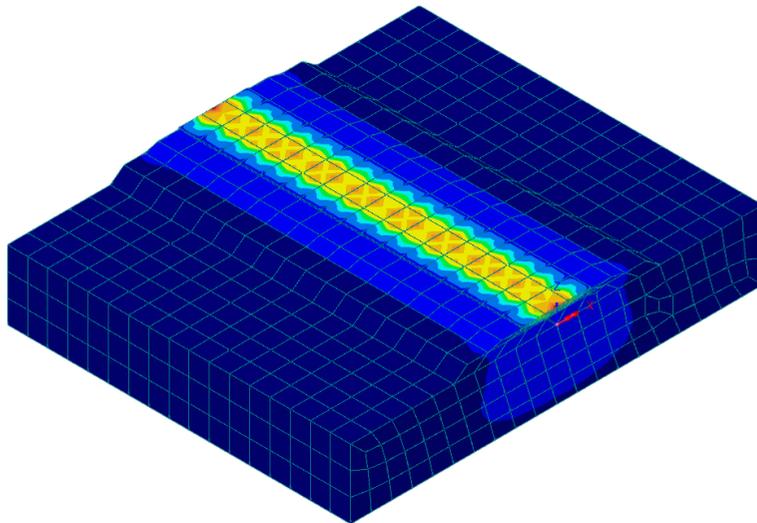
Figure 11: Envelope resultant displacement