

# Modelling of Moving Load on Embankment

For LUSAS version:	24.0
For software product(s):	LUSAS Bridge plus
With product option(s):	Geotechnical, Nonlinear

## Problem Description

This example studies 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 metres.

## Keywords

**Moving Load, Settlement.**

## Associated Files

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



**modelling\_of\_moving\_load\_on\_embankment.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.

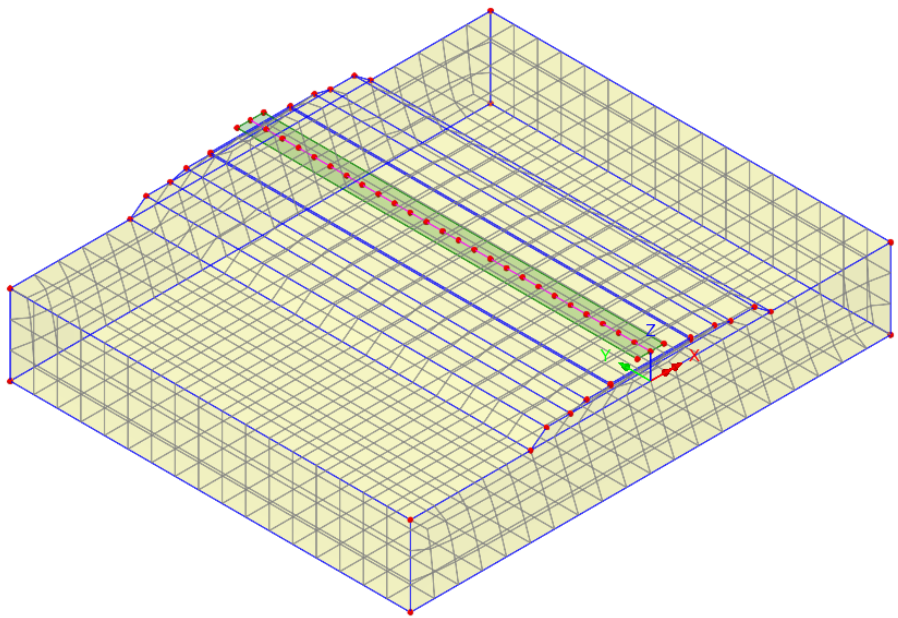
### Objectives

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.

### Modelling overview

The model is created using an analysis category of **3D** with model units of **kN,m,t,s,C**.

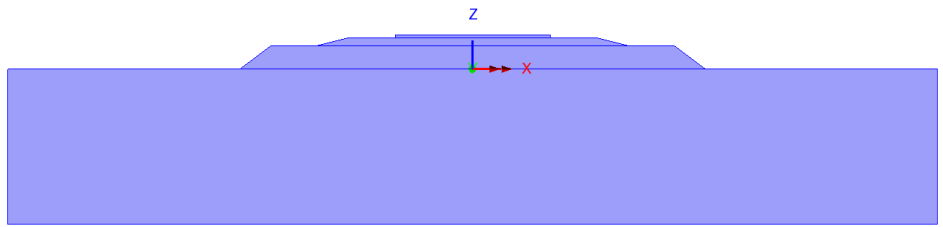
Running the supplied script, choosing an isometric view and turning off the display of the loading and supports will show this view of the model.



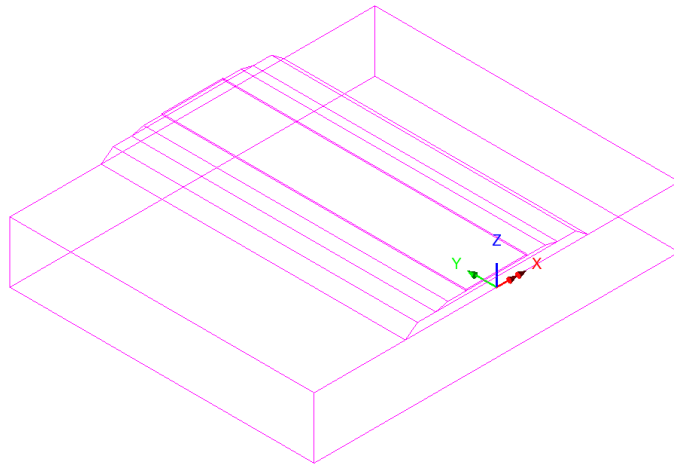
Model created from running the supplied script

### Feature Geometry

The model is created by defining surfaces in the XZ plane and then extruding them along the Y axis. It is essential to be diligent in ensuring that the surfaces are properly connected and that there are no overlaps.



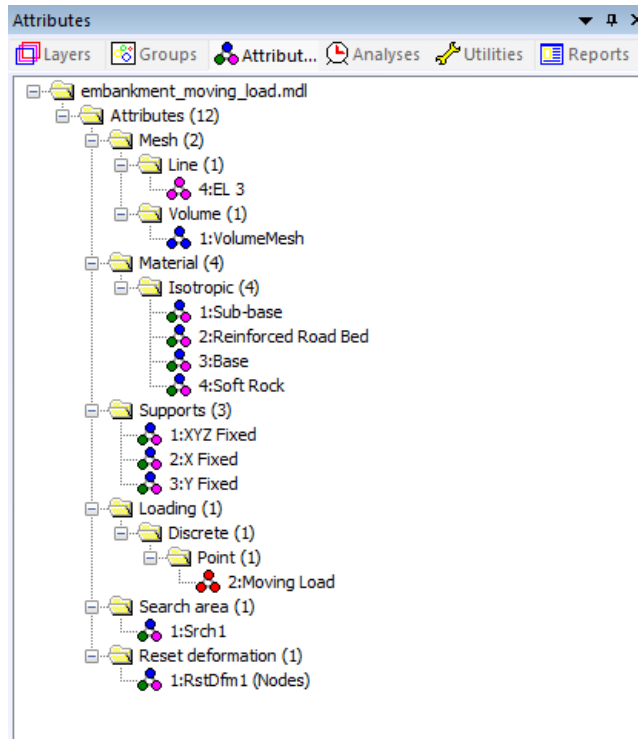
The model created in the XZ Plane



3D Model

## Attributes

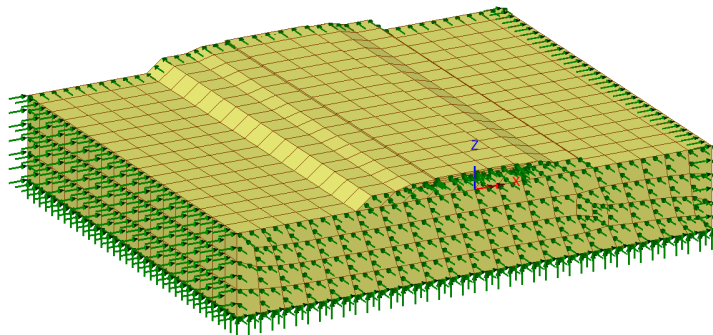
Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model.



Model Attributes

## Mesh

Hexahedral, quadratic stress elements (HX20) have been used for meshing.



Mesh and boundary conditions

## Materials

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

Table 1: material properties

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

## Supports

The model is restrained in X and Y direction, and fully fixed supports are assigned to the base.

## Loads

In addition to gravity, point loads are also created which apply a concentrated force representing wheel loads. 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.

Point

Analysis category 3D

Arbitrary

Grid x 2  
y 2

Untransformed load direction

X  XYZ

Y  Surface normal

Z

Projection vector

Project in load direction

Project for prestress

X component 0.0

Y component 0.0

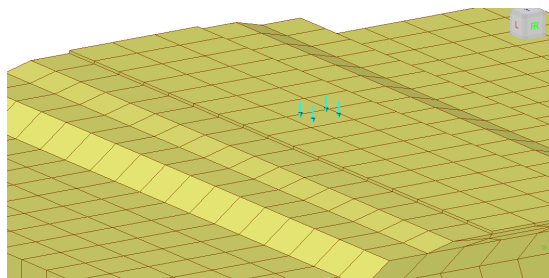
Z component 1.0

	X	Y	Z	Load
1	-0.75	-1.0	3.2	-150.0
2	-0.75	1.0	3.2	-150.0
3	0.75	1.0	3.2	-150.0
4	0.75	-1.0	3.2	-150.0

Name Moving Load (2)

Close Cancel Apply Help

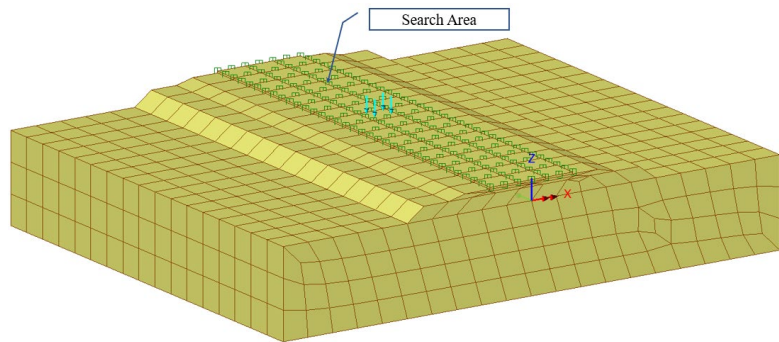
**Discrete point loading dialog**



**Discrete point loading**

### 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**.



Search area used to assign discrete point loading



**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.

The deformation in the first loadcase is reset using the **Reset Deformation** attribute.

## Analyses considered

### 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 with all parameters left as their default values.

### Moving Load Phases

In LUSAS, the moving load generator is used to create the load cases required to model a vehicle moving along a defined path. To access this facility, select the path of the moving load (line 93 in this example) before using the menu item **Bridge > Moving Load Analysis**, to bring up a dialog on which parameters such as 'Search area' (defined previously), 'Reference path' and 'Path divisions' can be stated and defined.

## Modelling of Moving Load on Embankment

---

Moving Load Analysis

Discrete load: 2:Moving Load

Assignment

Project onto line (2D line beams and frames)

Project over area (grillages, shells, and 3D space frames)

Search area: 1:Srch1

Options for loads outside search area: Include Full Load

Moments to include: None

Load path definition

Reference path: Create reference path from selection

Direction of travel: Forward

Path divisions

Move load along path in equal divisions Divisions: 26

Move load by incremental distance along path Distance: 1.92308

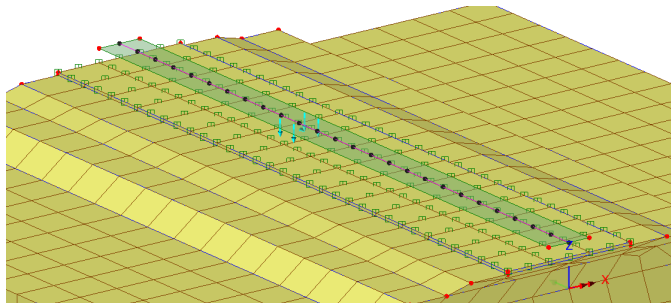
Create loadcase envelope

Name: Static Moving Load Analysis 1 (new)

OK Cancel Apply Help

**Moving Load Analysis dialog**

When done, 26 stages will represent the movement of the load along the path.



**Moving load path defined**

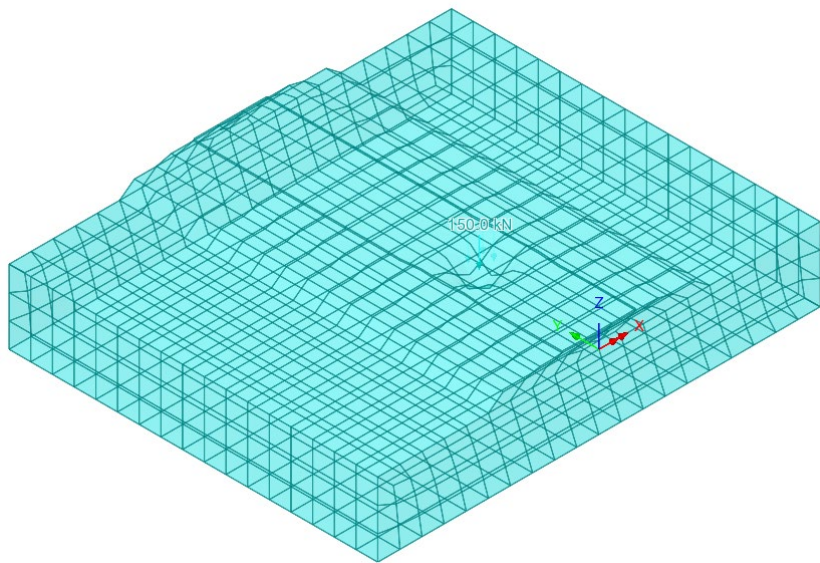
## Viewing the results

Analysis loadcase results are present in the Treeview.

- Turn off the display of the geometry and mesh layers.
- Set active loadcase 11 (load position 9)

### Deformed Mesh

The deformed mesh at loadcase 11 (load position 9) is shown below.



Deformed mesh at load position 9

### Settlement

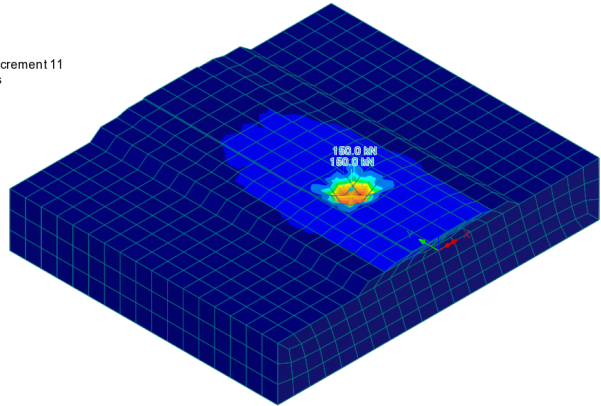
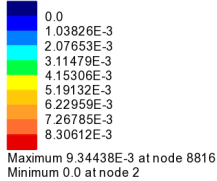
Since soil layers are homogenous, it is expected that the displacement along the moving load's path will be consistent with a value of around 10mm as shown below.

To see this:

- Add the contours layer to plot contours of entity **Displacement** and component **RSLT**

# Modelling of Moving Load on Embankment

Analysis: Analysis 1  
Loadcase: 11 LoadID=2 Line=93 Dir=Fwd Pos=9, 11, Increment 11  
Results file: embankment\_moving\_load-Analysis 1.mys  
Entity: Displacement  
Component (Nodal): RSLT (Units: m)



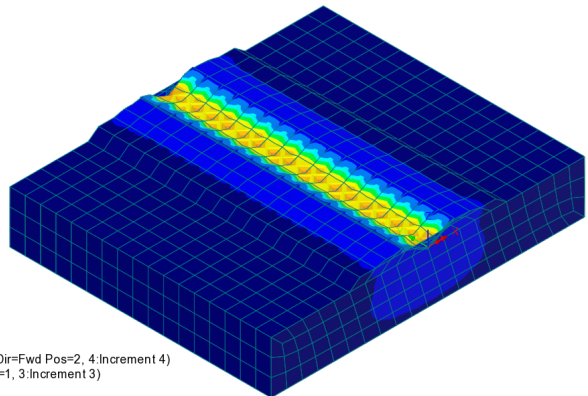
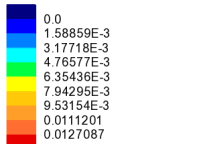
Contours of resultant displacement at load position 9

## Envelope

An overview of the maximum displacement along the path of the vehicle can be obtained by enveloping loads along that path. This is done by selecting the menu item **Analysis > Envelope** and including loadcases from 3 to 28 (positions 1 to 26). An envelope has already been created in the supplied model using these settings.

- Set active **Envelope 1** choosing **Maximum values** to see resultant displacements for the moving load across the embankment.

Envelope: 29 Envelope 1 (Max - RSLT)  
Entity: Displacement  
Component (Nodal): RSLT (Units: m)



Envelope of resultant displacements for all load positions.