Chapter 10
3D Moving Train Load Time History

Workflow

	: Main Menu > Import > DXF 2D(Wireframe)...	6
	: Geometry > Point & Curve > Intersect	6
	: Mesh > Control > Size Ctrl.	6
	: Mesh > Control > Match Seed	7
	: Mesh > Generate > 2D	7
	: Mesh > Protrude > Extrude	8
	: Mesh > Element > Create	9
	: Analysis > Analysis Case > General	9
	: Analysis > Analysis > Perform	9
	: Dynamic Analysis > Load > Train Dynamic Load Table	11
	: Mesh > Element > Create	12
	: Analysis > Analysis Case > General	12
	: Analysis > Analysis > Perform	13

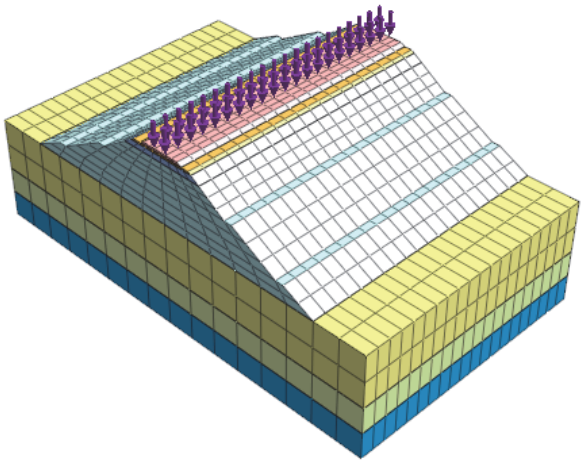
3D Moving Train Load Time History

Section 1
Overview

1.1 Learning Purpose

Train vibration is periodic loading phenomenon, which occurs as a result of train wheels passing above the ground surface. The period is related to axle interval and train velocity.
The characteristics of train vibration are affected by various factors such as vehicle, rail track, supporting structures, ground, submerged structure etc. These factors are interactively applied, generated, and spread. And it appears as complex vibration phenomenon.

▶Moving Train Load Time History

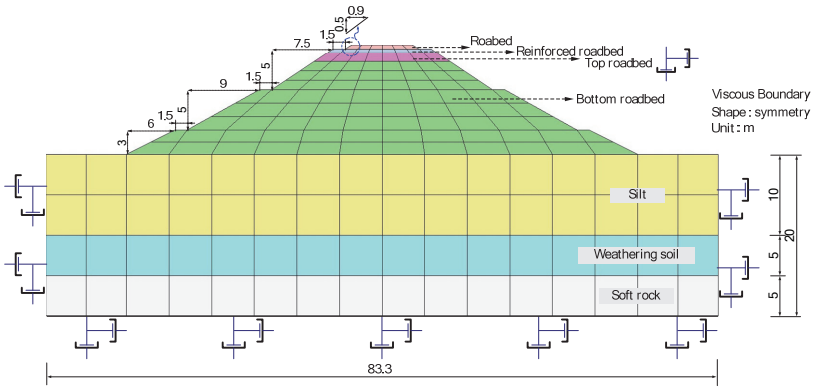


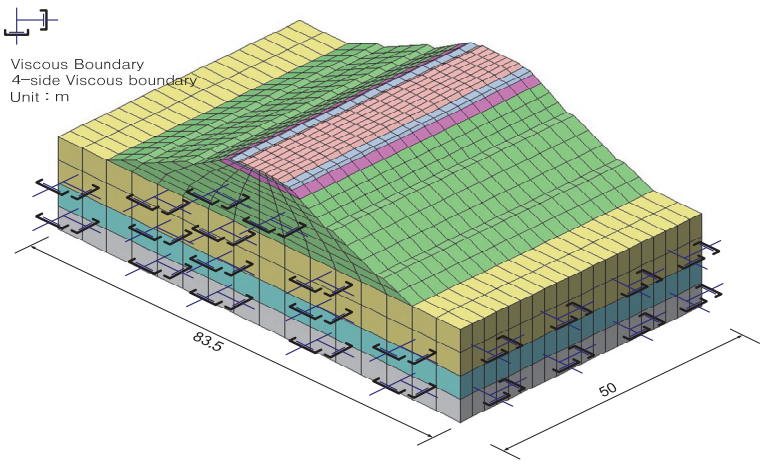
In this tutorial, the following main concepts will be explained:

- Generate 3D meshes by extruding from 2D meshes
- Eigenvalue analysis
- Generate moving train load
- Analysis result – surrounding vibration effect and vertical ground settlement.
- Analysis result – Create time history curve

1.2 Model and Analysis Summary

By performing dynamic analysis, this tutorial analyzes vibration effect on surrounding structures and ground behavior on condition that train moving load is applied to the embankment
Generate embankment as bottom roadbed, top roadbed, and reinforced roadbed on tree different ground layer. And finally create roadbed on top.





Section 2

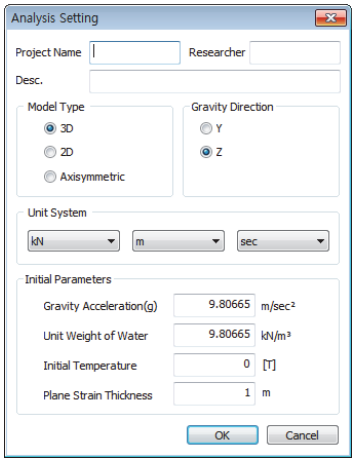
Analysis Setting

[Open the attached start file (10_train_start)]

∗ : Analysis > Analysis Case > Setting

- Set model type, gravity direction, initial parameters and unit system for the analysis. The unit system can be changed whenever you want during the modeling process and after performing the analysis. The input parameters will be automatically converted by the right unit system.
- This tutorial is a 3D model with gravity in Z direction and using SI unit system (kN, m, sec).

▶Setting analysis condition



Section 3
Define Material and Property

3.1 Definition of Ground and Structural Materials

For material modal type, apply 'Mohr-Coulomb' for the ground and 'Elastic' for the roadbed, which doesn't consider Nonlinearity.

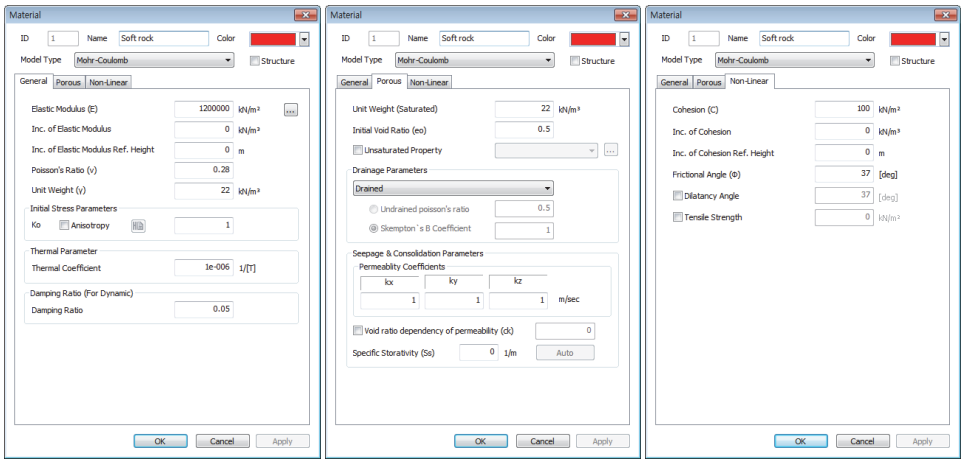
Ground and structure materials are defined as below:

▶Table. Ground material

[Unit : kN, m]

Name	Soft rock	Weathering soil	Silt	Roadbed_ Bottom	Roadbed_ Top	Reinforced roadbed	Roadbed
Material	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic
Model Type	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Elastic
General							
Elastic Modulus(E)	1.2E+06	2.0E+04	2.0E+04	1.0E+05	3.0E+04	1.3E+05	2.3E+07
Poisson's Ratio(v)	0.28	0.28	0.35	0.30	0.35	0.25	0.18
Unit Weight(r)	22	20	18	20	19	19	25
Ko	0.5	0.5	0.5	0.5	0.5	0.5	0.5
Porous							
Unit Weight(Saturated)	22	20	18	20	19	19	25
Initial Void Ratio(e0)	0.5	0.5	0.5	0.5	0.5	0.5	0.5
Drainage Parameters	Drained	Drained	Drained	Drained	Drained	Drained	Drained
Non-Linear							
Cohesion(C)	100	20	10	0	15	0	-
Frictional Angle	37	30	28	40	31	35	-

▶Define ground material - General
▶▶Define ground material - Porous
▶▶▶Define ground material – Non-linear



3.2 Definition of Properties

Properties represent physical attributes of the meshes and will be assigned to mesh sets during mesh generation. While defining ground and structure properties, firstly choose the material to be used. And for structure properties, structure types and cross-section shapes (cross-section stiffness) should be further defined.

▶Table. Ground property

Name	Soft rock	Weathering soil	Silt	Roadbed _ Bottom	Roadbed _ Top	Reinforced roadbed	Roadbed	Plot Only
Type	3D	3D	3D	3D	3D	3D	3D	Plot Only(2D)
Material	Soft rock	Weathering soil	Silt	Roadbed _ Bottom	Roadbed _ Top	Reinforced roadbed	Roadbed	-

Tip

'Plot Only' meshes are 2D meshes of ground/embankment which are created to be extruded to 3D model in later steps. Therefore no section or material information is needed.

Section 4
Modeling

The main purpose of this tutorial is to study extruding 3D meshes from 2D meshes, defining moving train load, and verifying results. You can start the tutorial by opening the start file in which basic materials and properties have already been predefined.

4.1 Modeling Geometry

∗ : Main Menu > Import > DXF 2D(Wireframe)...

Import DXF file of over AutoCAD R13 version.

- Open '10_train.dxf'.

∗ : Geometry > Point & Curve > Intersect

Perform this step to create nodes at the intersections.

- Select all lines. Click [OK].

4.2 Generate Mesh

∗ : Mesh > Control > Size Ctrl.

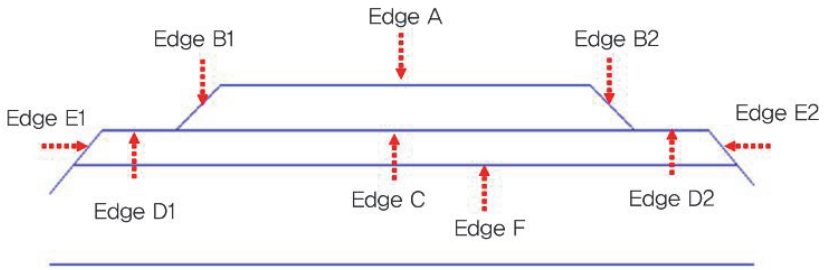
The mesh size has to be controlled to get high quality meshes with less number. This can be done easily using the size control.

- Refer to the image below, Select 'Edge B1, B2, D1, D2, E1, E2, G1, G2, K1, K2, N1, N2' of embankment.
- Choose method 'Number of Divisions'. Input '1'.
- Select Preview button to check the generated edge seeds. Press [Apply].
- Refer to the table below for mesh seeds at edges.

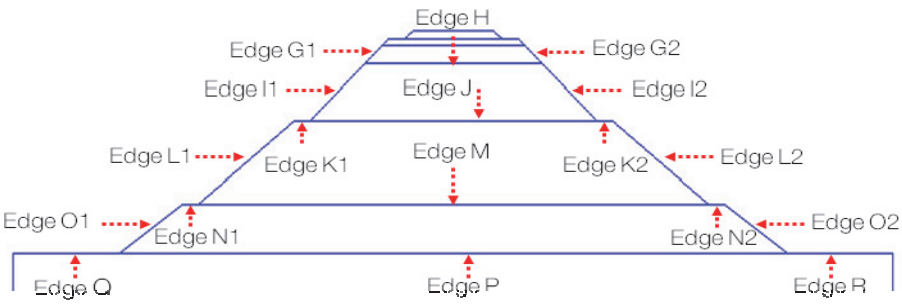
▶Table. Size control method and divisions.

Edge	Method	Division
B1, B2, D1, D2, E1, E2, G1, G2, K1, K2, N1, N2	Number of Divisions	1
I1, I2, L1, L2	Number of Divisions	3
O1, O2	Number of Divisions	2
A, C	Number of Divisions	6
F, H, J	Number of Divisions	8
M	Number of Divisions	10
P	Number of Divisions	12
Q, R	Number of Divisions	2

▶Edge number(A~F)



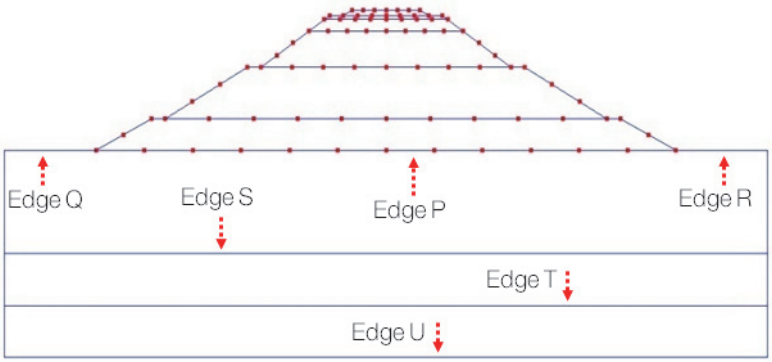
▶Edge number(G~R)



Tip

Through checkbox of Work Tree > Mesh Control > Mesh Size, it is possible to see generated seeding of the geometry shape.

▶Edge number(Q~U)



Use [Match Edge Seed] to define mesh sizes for the rest part based on existing seeding information

∗ : **Mesh > Control > Match Seed**

- Refer to upper image, select Target edge 'Edge S'.
- Select source edge 'Edge P, Q, R'.
- Choose match method to 'Projection'.
- Select Preview button to check the generated edge seeds. Press [Apply].
- In the same way assign seeding to 'T','U'.

Use Map mesh generation algorithm to created 2D mesh.

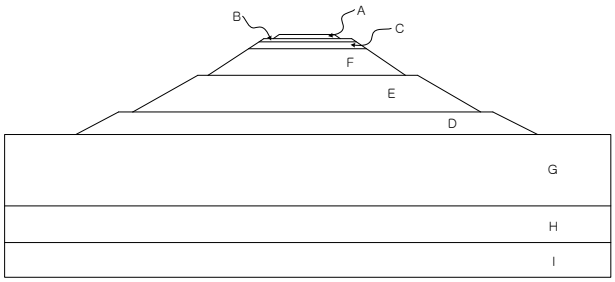
∗ : **Mesh > Generate > 2D**

- Select [Map-Area] tab.
- Select 'Automatic Mapped Boundaries'. And choose 4 edges which consist of each area se shown image below.
- Input size to '5' and choose property to '8: Plot only'.
- Input mesh set name to 'Roadbed (2D)'.
- Click [Apply].
- In the same way, create 2D mesh for areas B~I.

▶Table. Mesh set name

Area	Mesh set name
A	Roadbed(2D)
B	Reinforce roadbed(2D)
C	Roadbed_ Top(2D)
D, E, F	Roadbed_ Bottom(2D)
G	Silty(2D)
H	Weathered rock(2D)
I	Soft rock(2D)

▶Generated mesh(2D)



Tip

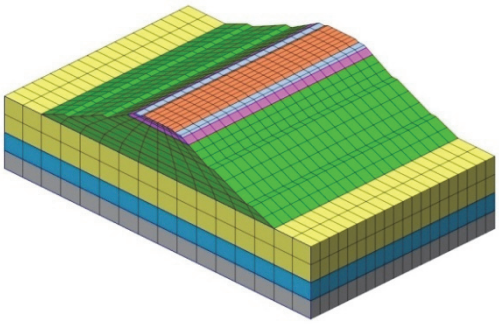
If seeding value is already assigned to an object, mesh size inputted to generate mesh for this object will not come into effect. Instead seeding value will be applied as a priority. Although mapped mesh isn't the most convenient way to process modeling, it has advantage of generating meshes of higher quality.

* : Mesh > Protrude > Extrude

Extrude 2D meshes along Y direction to generate 3D meshes.
Divide total length of 50m into 20 for mesh generation

- Select [2D->3D] tab.
- Set [Selection filter] to 'Element(T)'. Select 'Soft rock (2D)'.
- The elements of soft rock which is source for extruding will be not used in further steps, check 'Delete' option to delete afterwards.
- Choose 'y' as extrude direction
- Set [Extrude information] to 'Uniform', 'Offset/Times'.
- Input '2.5', '20' to 'Offset/Times'.
- Assign property 'Soft rock'.
- Name the mesh set as 'Soft rock'. Press [Apply].
- In the same way, create 3D meshes for Weathering soil', 'Silt', 'Roadbed_Bottom', 'Roadbed_Top', 'Reinforced roadbed', 'Roadbed'.

▶Generated mesh(3D)



Section 5
Analysis Setting (Eigenvalue)

Case 1
Eigenvalue analysis is used to analyze dynamic property of structure itself. It is also called 'Free Vibration Analysis. This tutorial determines the damping matrix which will be used in time history analysis by calculating nature periods of first and second modes in Eigenvalue analysis.

5.1 Setting Boundary Condition

∗ : **Mesh > Element > Create**

- In the [Other] tab, choose 'Ground Surface Spring'.
- Select all the mesh set.
- Select elastic boundary, and input 'Modulus of Subgrade Reaction' to '1'.
- Check [Fixed Bottom Condition] option, name mesh set as 'Elastic boundary'.
- Click [OK].

Tip

To perform Eigenvalue analysis, define point condition using elastic boundary. Calculate spring boundary value according to ground reaction coefficient of Railroad Design Criteria.

Vertical ground reaction coefficient : $k_v = k_{v0} \cdot (\frac{B_v}{30})^{-3/4}$

Horizontal ground reaction coefficient : $k_h = k_{h0} \cdot (\frac{B_h}{30})^{-3/4}$

Here, $k_{vo} = \frac{1}{30} \cdot \alpha \cdot E_0 = k_{h0}$, $B_v = \sqrt{A_v}$, $B_h = \sqrt{A_h}$

Av and Ah are cross sections in vertical and horizontal directions. E0 is ground elastic modulus. α is usually applied as 1.0.
In GTS NX, it is possible to generate elastic boundary easily through ground surface spring.

5.2 Setting Analysis Case

∗ : **Analysis > Analysis Case > General**

- Name the analysis case as 'Eigenvalue'.
- Select solution type to 'Eigenvalue'.
- Move all the mesh sets and boundary conditions to Active Sets.
- Click [OK].

5.3 Perform Analysis

∗ : **Analysis > Analysis > Perform**

- Perform analysis

Section 6
Results (Eigenvalue)

Post-processing starts after the analysis completes normally. Result from Eigenvalue analysis is not the final result. Our final purpose is to evaluate the ground behavior and response in dynamic analysis. This step is just to obtain the value needed in further analysis.

∗ : **Result > Eigenvalue > Mode Analysis Result Table**

- Double click mode analysis result table, and check the values of first and second period modes, in which mass participation rates are high.

▶Mode analysis result table

REAL EIGENVALUES								
MODE NUMBER	EIGENVALUE	RADIANS	CYCLES	PERIOD	GENERALIZED MASS	GENERALIZED STIFFNESS	ORTHOGONALITY LOSS	ERROR MEASURE
1	1.486319e+001	3.855281e+000	6.135871e-001	1.629760e+000	1.000000e+000	1.486319e+001	0.000000e+000	6.386602e-012
2	1.961503e+001	4.428885e+000	7.048790e-001	1.418683e+000	1.000000e+000	1.961503e+001	0.000000e+000	5.503301e-012
3	2.542789e+001	5.042607e+000	8.025559e-001	1.246019e+000	1.000000e+000	2.542789e+001	0.000000e+000	1.958028e-012
4	4.704788e+001	6.859146e+000	1.091667e+000	9.160303e-001	1.000000e+000	4.704788e+001	0.000000e+000	2.611241e-012
5	4.929095e+001	7.020752e+000	1.117387e+000	8.949451e-001	1.000000e+000	4.929095e+001	0.000000e+000	5.073442e-012
6	6.349148e+001	7.968154e+000	1.268171e+000	7.885372e-001	1.000000e+000	6.349148e+001	0.000000e+000	2.105789e-012
7	6.669329e+001	8.166596e+000	1.299754e+000	7.693764e-001	1.000000e+000	6.669329e+001	0.000000e+000	1.507926e-012
8	6.899587e+001	8.306375e+000	1.322001e+000	7.564291e-001	1.000000e+000	6.899587e+001	0.000000e+000	3.405623e-010
9	7.109971e+001	8.432064e+000	1.342005e+000	7.451537e-001	1.000000e+000	7.109971e+001	0.000000e+000	2.443518e-011
10	7.603368e+001	8.719729e+000	1.387788e+000	7.205712e-001	1.000000e+000	7.603368e+001	0.000000e+000	3.164651e-009

MODAL EFFECTIVE MASS								
MODE NUMBER	T1	T2	T3	R1	R2	R3		
1	0.000000e+000	1.051802e+008	0.000000e+000	1.420140e+010	0.000000e+000	0.000000e+000		
2	1.263321e+008	0.000000e+000	0.000000e+000	0.000000e+000	1.063867e+010	0.000000e+000		
3	0.000000e+000	0.000000e+000	0.000000e+000	0.000000e+000	0.000000e+000	7.965364e+010		
4	0.000000e+000	2.379728e+007	0.000000e+000	3.749985e+009	0.000000e+000	0.000000e+000		
5	0.000000e+000	0.000000e+000	7.387765e+007	0.000000e+000	0.000000e+000	0.000000e+000		
6	0.000000e+000	6.560521e+005	0.000000e+000	1.049821e+010	0.000000e+000	0.000000e+000		
7	2.069277e+006	0.000000e+000	0.000000e+000	0.000000e+000	2.074238e+010	0.000000e+000		
8	0.000000e+000	0.000000e+000	0.000000e+000	1.557506e-010	4.605718e-011	5.828123e+009		
9	0.000000e+000	0.000000e+000	1.047309e+007	1.065030e-012	0.000000e+000	0.000000e+000		
10	2.650355e+005	2.045152e-012	0.000000e+000	2.245241e-008	1.838852e+009	0.000000e+000		
TOTAL	1.286664e+008	1.296335e+008	8.435075e+007	2.844960e+010	3.321990e+010	8.548176e+010		
TOTAL IN MODEL	2.137232e+008	2.137232e+008	2.137232e+008	6.050075e+010	1.204230e+011	1.494188e+011		

PERCENTAGE MODAL EFFECTIVE MASS								
MODE NUMBER	T1	T2	T3	R1	R2	R3		
1	0.00%	49.21%	0.00%	23.47%	0.00%	0.00%		
2	59.11%	0.00%	0.00%	0.00%	8.83%	0.00%		
3	0.00%	0.00%	0.00%	0.00%	0.00%	53.31%		
4	0.00%	11.13%	0.00%	6.20%	0.00%	0.00%		
5	0.00%	0.00%	34.57%	0.00%	0.00%	0.00%		
6	0.00%	0.31%	0.00%	17.35%	0.00%	0.00%		
7	0.97%	0.00%	0.00%	0.00%	17.22%	0.00%		
8	0.00%	0.00%	0.00%	0.00%	0.00%	3.90%		
9	0.00%	0.00%	4.90%	0.00%	0.00%	0.00%		
10	0.12%	0.00%	0.00%	0.00%	1.53%	0.00%		
TOTAL	60.20%	60.65%	39.47%	47.02%	27.59%	57.21%		

Section 5
Analysis Setting (Time History Analysis)

Case2
5.1 Setting Load Condition (Train dynamic load)

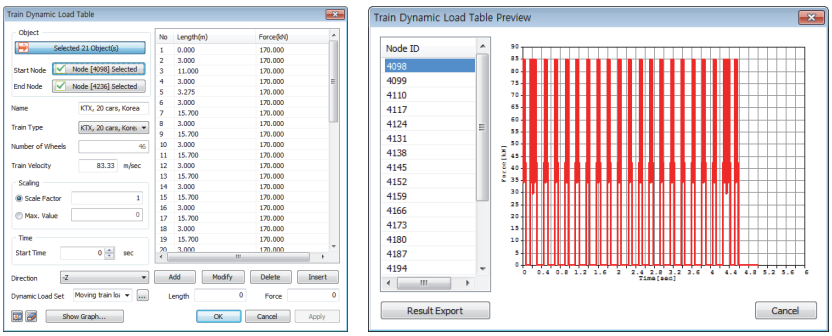
Open '10_train_model.gts' which only consists of 3D Mesh Model right before assigning boundary conditions. Set boundary condition and load condition in this file for train dynamic load analysis. Perform time history analysis.
In static analysis, when analyzing a model with infinite material such as ground, boundaries are set far enough from main analysis area. But in dynamic analysis since effect of waves reflection occurs, if boundaries are set in the same way as static analysis, big error may occur. To prevent errors, we use approximate methods, such as artificial boundary, transmitting boundary/ hyper element, infinite element, and boundary element etc. In this analysis 'Viscous boundary and quiet boundary' by Lysmer and Wass are applied.

* : Dynamic Analysis > Load > Train Dynamic Load Table

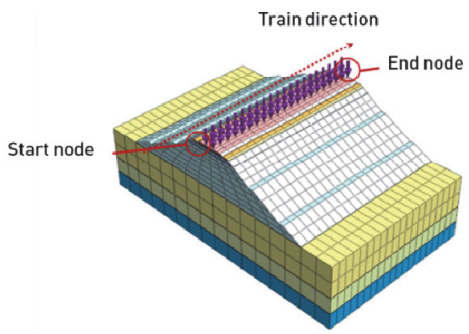
Apply train dynamic load condition to the model. Is it advisable to use real data for time-load function, this example uses load which simulates KTX20 cars standard (300km/h=83.33m/s) by moving force model.
Moving forcing model assumes that when the moving train passes through each node of the model in a short period, impulsive load is applied to the nodes. And this impulsive load is idealized as form of triangle.

- Select nodes (Where the wheel passes) to which train load will applied
- Select start node and end node by considering train direction.
- Set the train type to 'KTX, 20 cars', input train velocity to 83.33m/sec.
- Choose direction to '-Z'.
- Name dynamic load set as 'Train dynamic load'.
- Since rain wheels are located at both left and right; generate one more dynamic load function with same start, end nodes on different target object.
- Press [Show Graph] and see the dynamic load function at the each node.

▶Train dynamic load table
▶▶Train dynamic train load function



▶Define train dynamic load



5.2 Setting Boundary Conditions

∗ : Mesh > Element > Create

- Select 'Ground Surface Spring' in [Other] tab.
- Select all mesh sets.
- Check [Damping Constant/Area].
- Check [Fixed Bottom Condition]. Enter mesh set name to 'Viscous boundary'.
- Click [OK].

Tip

To define viscous boundary, calculate and input damper value about x, y, z direction according to ground material. Formulas to calculate Damper value are shown below.

About P wave, $C_p = \rho \cdot A \cdot \sqrt{\frac{\lambda + 2G}{\rho}} = W \cdot A \cdot \sqrt{\frac{\lambda + 2G}{W \cdot 9.81}} = c_p \cdot A$

About S wave, $C_s = \rho \cdot A \cdot \sqrt{\frac{G}{\rho}} = W \cdot A \cdot \sqrt{\frac{G}{W \cdot 9.81}} = c_s \cdot A$

Here, $\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)}$, $G = \frac{E}{2(1+\nu)}$, E : Elastic modulus, ν : Poisson's ratio, A : Area

In GTS NX, it is possible to generate viscous boundary easily through 'Ground surface spring'.

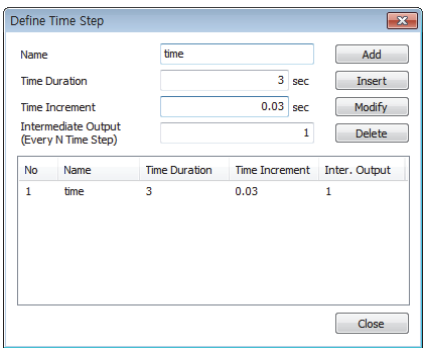
5.3 Setting Analysis Case

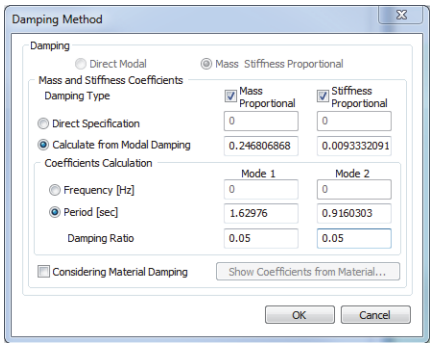
∗ : Analysis > Analysis Case > General

- Name the case as 'Train dynamic load'.
- Select solution type 'Linear Time History (Direct).
- In [Time Step] press define step. Input time which will reflect to the analysis.
- Enter name. Input [Time Duration] to '3' and [Time Increment] to '0.03'.
- In [Analysis control], define damping.
- Choose 'Calculate from Modal Damping', put the value of first, second period which is from [5-1. Analysis (Eigenvalue)].
- Enter '0.05'.
- Click [OK] and close analysis control window.
- All the mesh set, boundary condition, load condition to the [Active Sets].
- Click [OK].

▶ Define time step

▶▶ Input damping mode





5.4 Perform Analysis

∗ : Analysis > Analysis > Perform

- Perform analysis.

Tip

During the analysis, you can check the calculation process in real-time. Messages such as whether the results converge or not, warnings and errors can be checked through [Output Window].
The model is automatically saved before being analyzed. The information about analysis is saved in *.OUT file in the same place where the model file is saved.
When analyzing a model, the results can be controlled by Analysis > Analysis Case > General > Output Control option. And if you set the output option as [Binary and Text], the results of nodes and elements after the analysis are also plotted in *.OUT file.

Section 6
Results

After the analysis, it is possible to verify deformation, stress results in the result tree. In this tutorial, the main result items to be checked are listed below.

- Displacement/Acceleration form according time
- Stress of entire time history

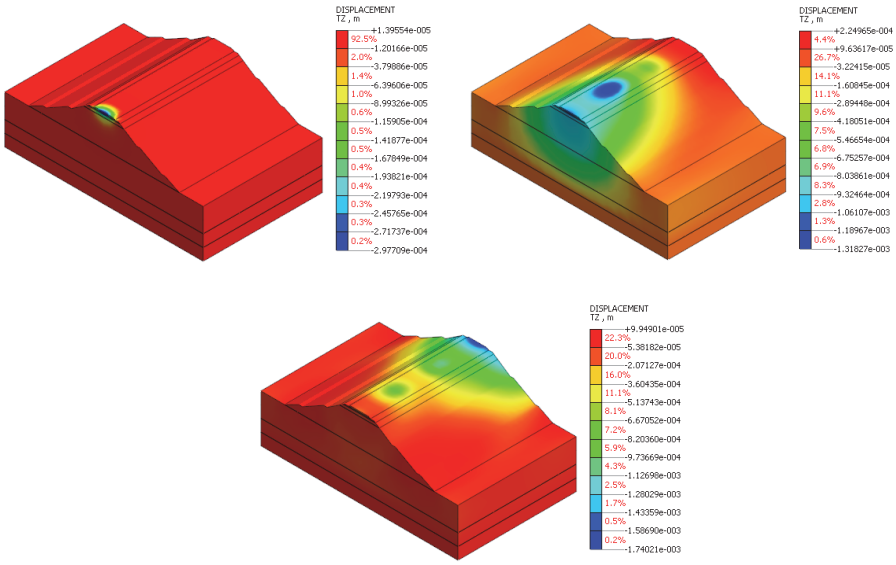
6.1 Displacement/Acceleration according to time

In Time history analysis results of MIN, MAX, and ABSOLUTE MAX are plotted. MIN plots result at the point where minimum value is generated; MAX represents maximum value, while ABSOLUTE MAX represents absolute maximum value.
Use ABOSULTE MAX to verify displacement.

- Select Work Tree > Results > Train dynamic load > Linear Time History (Direct) > ABSOLUTE MAX > Displacements > TOTAL TRANSLATION (V).

- Scroll on the movement bar at the bottom of the screen to view the analysis results stage by stage.

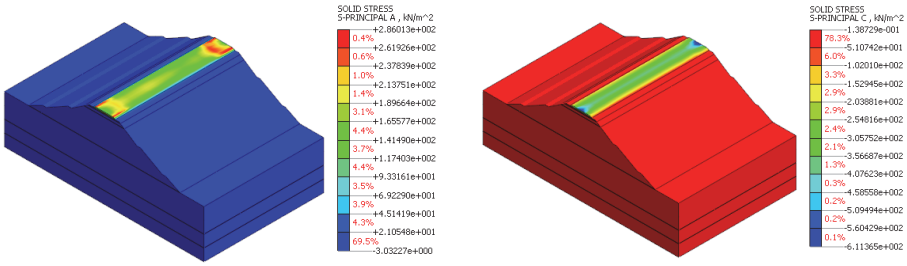
▶Displacement at T=0.03sec
▶▶Displacement at T=0.45sec
▶▶▶Displacement at T=0.09sec



6.2 Stress of entire time history

Stresses generated at the ground can be verified at 'Solid Stresses' in the Results Tree. S-XX, S-YY, S-ZZ represents stresses of each direction, maximum principal stress is S-PRINCIPAL A(V), and minimum principal stress is S-PRICIPAL C(V).

- Select Work Tree > Results > Train Dynamic Load > Linear Time History (Direct) > ABSOLUTE MAX > Solid Stresses > S-PRINCIPAL A (V), S-PRINCIPAL C (V). Verify maximum, minimum principal stress of the entire time history.

▶Maximum principal tress

▶▶Minimum principal stress