Dynamics: Data Preparation

August 2011

Dynamics: Scope of the lecture

- Tools for dynamic analyses in Z_Soil
  1. Eigenvalue and eigenmodes detection
  2. Time history analysis
  3. Added masses
  4. Seismic input
  5. Boundary conditions
  6. Periodic boundary conditions
  7. Initial conditions
  8. Viscous boundaries
  9. Domain Reduction Metod (DRM)

This lecture is devoted to structural and single-phase continuum transient dynamics. We will discuss basic dynamic drivers that allow you to solve a given initial boundary value problem. The main modeling aspects are considered including setting boundary conditions, initial conditions, viscous boundaries (for continuum), modeling 1D shear layer problem via 2D or 3D FE models. A special attention will be paid to the reduction of the size of computational model using so called Domain Reduction algorithm.

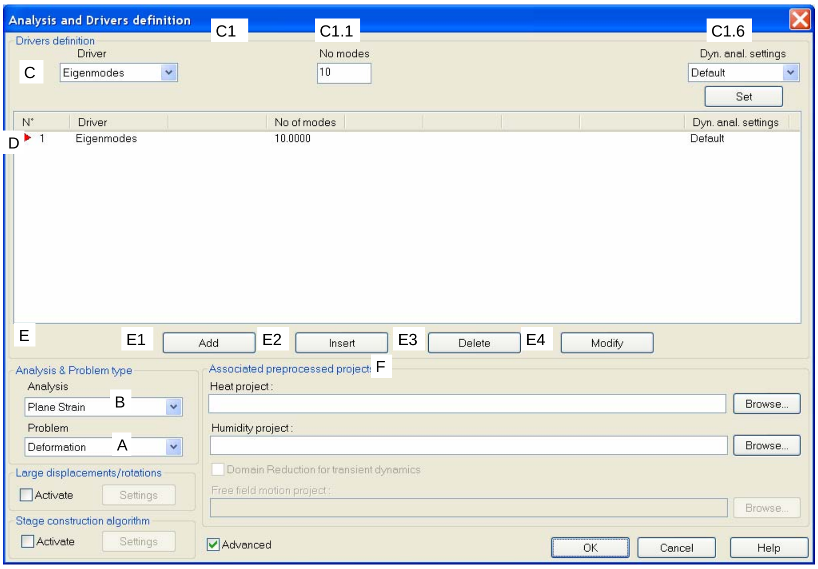
Dynamic drivers

1. Eigenvalue problem
2. Time history analysis for single-phase media
   (*Transient dynamics-Driven load*)
3. Time history analysis for two-phase media
   (*Transient dynamics-Consolidation*)

The eigenvalue and eigenmodes driver is added to the list of dynamic drivers to allow the user to calculate a selected set of free vibration modes and corresponding eigenfrequencies. The second driver is the true transient dynamic one that allows to carry out time history analysis for structures and single-phase/two-phase **(but uncoupled)** continuum. The third driver is designed to carry out time history analysis for two phase fully or partially saturated continuum including structures.

Dynamic driver: Eigenvalue

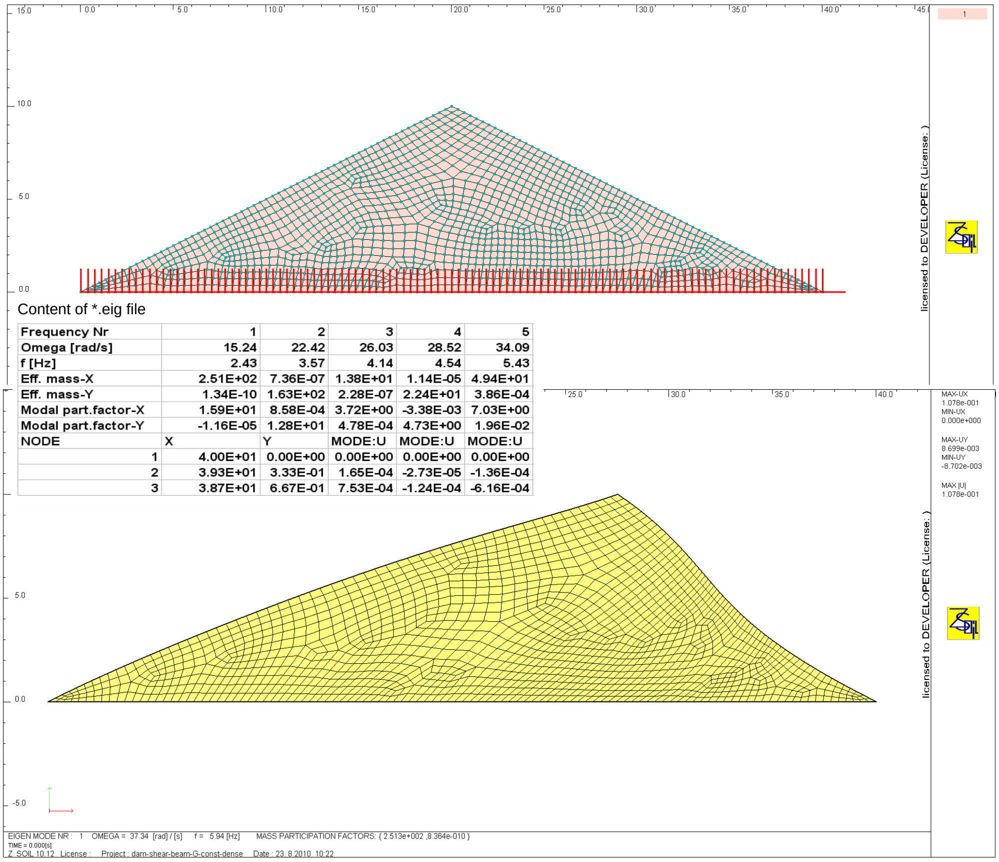
Analysis & Drivers menu



- Number of detected eigenvalues is limited by total number of free degrees of freedom
- For each eigenmode mass participation factors are computed for all directions
- Lanczos method is used

The eigenvalue driver produces eigenmodes and eigenfrequencies that can be visualized in the postprocessor bu running the option *Graph analysis/Eigenmodes*. Once we are in the postprocessor the selection of a given eigenmode can be done by using the combo-box placed next to the time selection one.

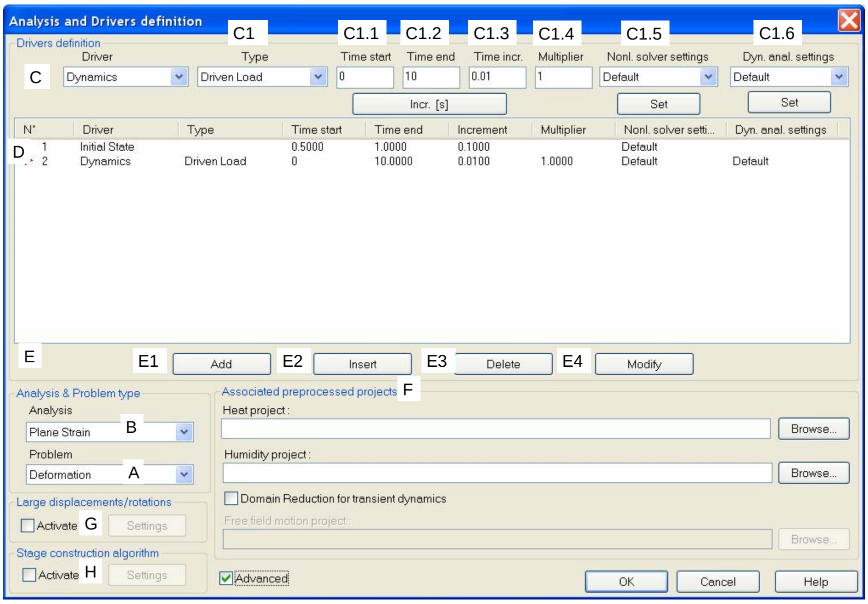
Eigenvalue and eigenmode analysis: example of dam



Frequency Nr	1	2	3	4	5
Omega [rad/s]	15.24	22.42	26.03	28.52	34.09
f [Hz]	2.43	3.57	4.14	4.54	5.43
Eff. mass-X	2.51E+02	7.36E-07	1.38E+01	1.14E-05	4.94E+01
Eff. mass-Y	1.34E-10	1.63E+02	2.28E-07	2.24E+01	3.86E-04
Modal part.factor-X	1.59E+01	8.58E-04	3.72E+00	-3.38E-03	7.03E+00
Modal part.factor-Y	-1.16E-05	1.28E+01	4.78E-04	4.73E+00	1.96E-02
NODE	X	Y	MODE:U	MODE:U	MODE:U
1	4.00E+01	0.00E+00	0.00E+00	0.00E+00	0.00E+00
2	3.93E+01	3.33E-01	1.65E-04	-2.73E-05	-1.36E-04
3	3.87E+01	6.67E-01	7.53E-04	-1.24E-04	-6.16E-04

The eigenfrequencies are output in the legend in form of ω [rad/s] and f [Hz]. In addition so called mass participation factors are computed that give us an information on the ammount of the mass of the structure that is involved in a given mode in given direction. This information is also stored in the *.eig file that can be visualized with Excel from main menu *Results/Eigenvectors*.

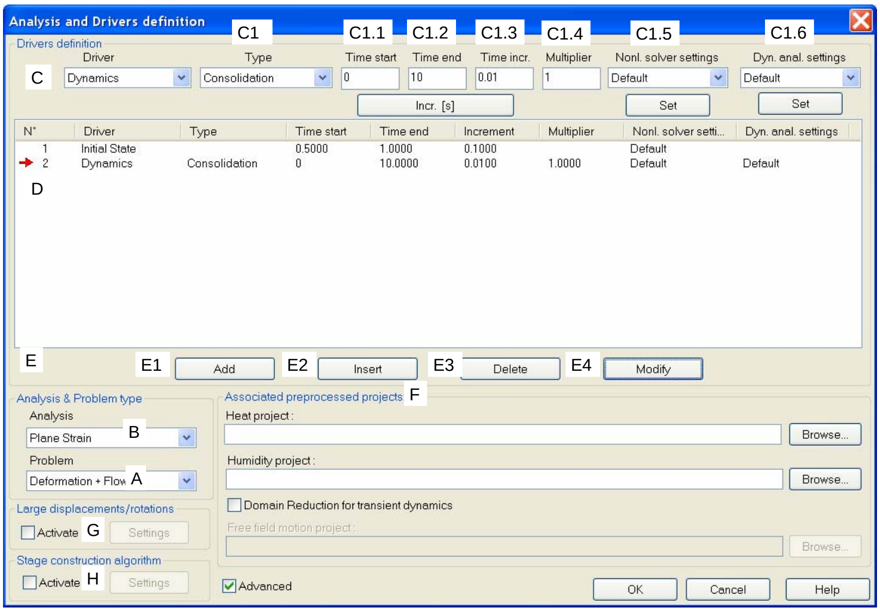
Dynamic driver: Dynamics: Driven load



1. This driver can be used in conjunction with *Deformation* (single phase problems) or *Deformation+Flow* (two-phase but uncoupled)
2. Newmark or HHT scheme can be used
3. HHT is recommended

If this driver is used in the *Deformation+Flow* mode the pore pressures will remain unchanged during dynamic analysis. At this level one may associate a free field solution with a given dynamic driver, once the Domain Reduction Method is used. Only one far field project can be used, no matter how many dynamic drivers are present in the list.

Dynamic driver: Dynamics: Consolidation

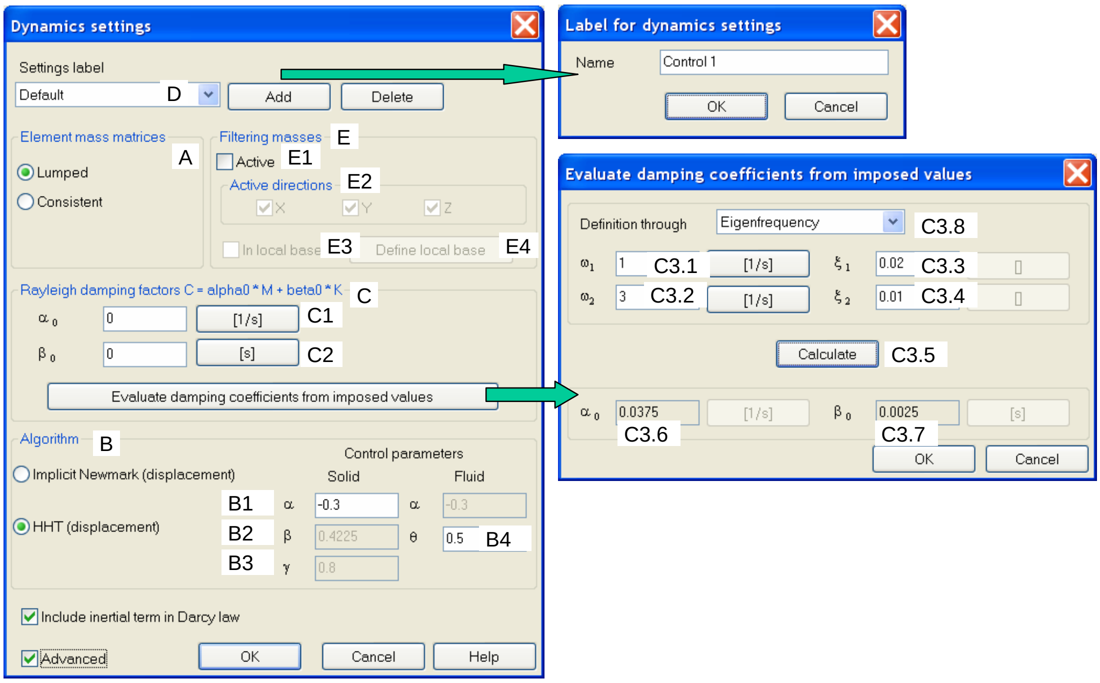


1. This driver can only be used in conjunction with *Deformation+Flow* (two-phase coupled)
2. Newmark or HHT scheme can be used
3. HHT is recommended

Same remarks as for the single-phase continuum apply to the dynamic driver that solves coupled dynamic consolidation problems. However the integration scheme for the pore pressure is the same as for the standard static consolidation (please refer to the dedicated report on dynamics for more details)

Dynamic drivers: Specific settings

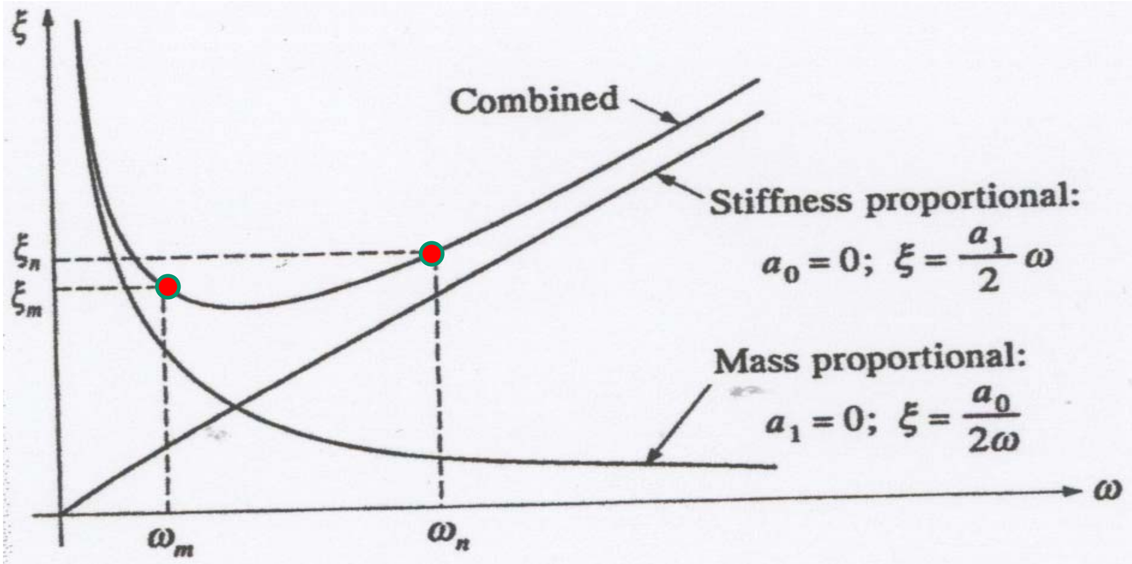
Control/Dynamics



1. Lumped/Consistent mass matrices can be used

This dialog box allows to set up one of the implicit integration schemes (Newmark or HHT), modify default integration coefficients (if needed), make a global seting for Raylaigh damping that can be overriden at the material level and impose mass filter in a given direction. The HHT integration method is highly recommended. It introduces some sort of numerical damping for high frequencies while Newmark scheme with $\beta = 0.25$ and $\gamma = 0.5$ yields zero damping. Application of Newmark scheme with parameters different from $\beta = 0.25$ and $\gamma = 0.5$ may be unsafe as it damps both high and low frequencies. The HHT parameters β and γ are automatically computed once the α is given (default $\alpha = -0.3$).

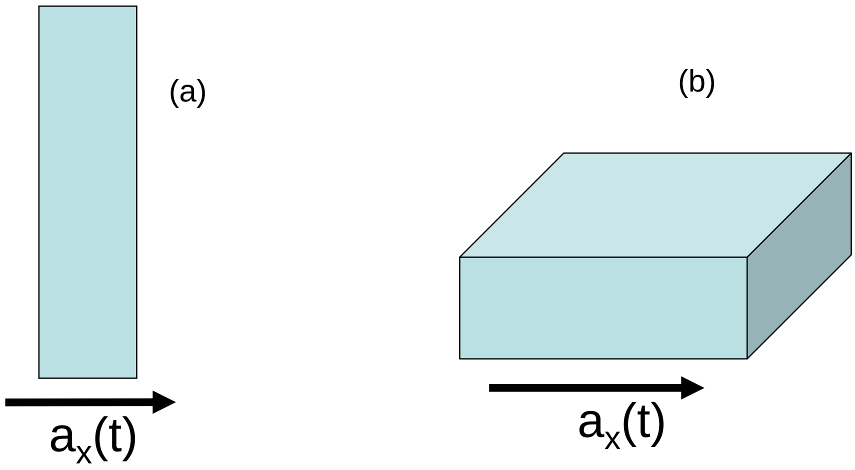
Specific settings: Rayleigh damping



Rayleigh damping is used to represent minor nonlinearities in soil/structure behavior that is not included when linear elasticity is used. As we can see it is frequency dependent. The mass proportional damping damps low frequencies while stiffness proportional high frequencies. The Rayleigh parameters set here are treated as a global setting inherited by all materials. To cancel or modify this setting one may do it at the material level in group ☒ **Damping**. In the current implementation stiffness proportional damping makes sense only for linear problems.

Specific settings: Filtering masses

Why do we need this option ?



1. Case (a): to cancel dilatational waves (shear layer mode)
2. Case (b): to cancel motion in the direction perpendicular to $a_x(t)$
3. This option applies as well to the added masses

Using this option one may easily cancel a motion in a given direction.

Requirements for time and space discretization

- To trace wave propagation in the medium we need approximately 10 nodes per wavelength
- The mesh size depends on the maximum frequency f_{max} that is to be represented
- For typical seismic application f_{max} is limited up to 10 Hz
- Hence the maximum mesh size should be smaller than

$$h^e \leq \frac{\lambda}{10} = \frac{v}{10\, f_{max}}$$

  where v is the lowest wave velocity that is to be considered (in most cases v is taken as shear wave velocity)
- Size of the applied time step, even for implicit integration schemes, is limited to a certain value too
- This is so due to the fact that the smallest fundamental period of vibration needs to be represented by at least 10 points (same amount as for the spatial discretization)
- Hence the time step limitation can be formulated as follows (v is the highest wave velocity.)

$$\Delta t \leq \frac{h^e}{v} \qquad (1)$$

As in the analysis we use a uniform time step hence meshes should satisfy the above requirements. In the two-phase applications we use $u - p$ formulation that works fairly well till 10 Hz.

Added masses

Definition

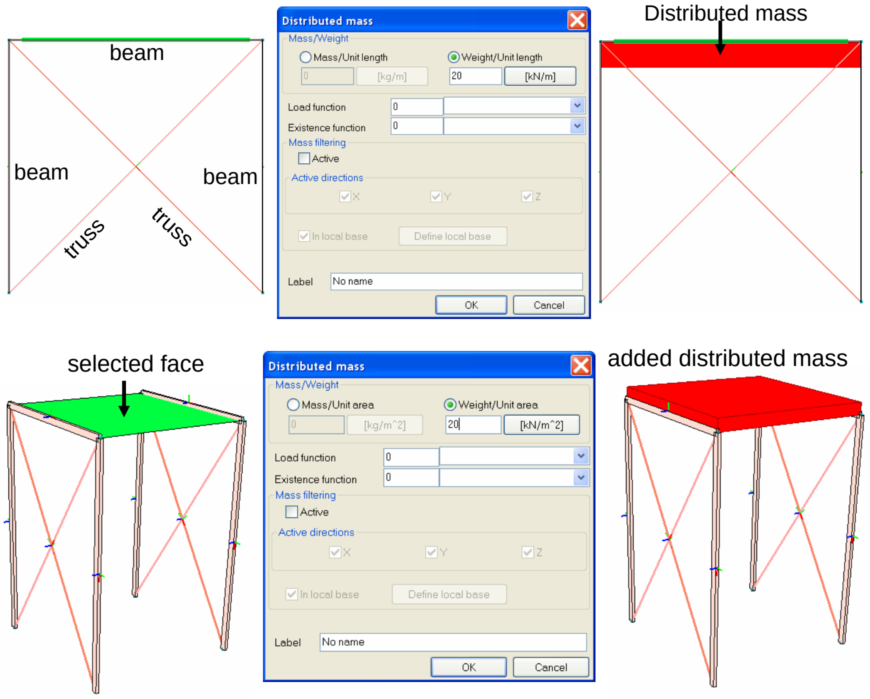
- 2D
  1. Nodal mass
  2. Mass distributed along the edge
- 3D
  1. Nodal mass
  2. Mass distributed along the edge
  3. Mass distributed on the face

Extended options

- Variation in time
- Existence function
- Filtering (can be useful for modelling added masses in dam-reservoir analysis)

Added masses can be defined on nodes, edges and faces of structural or continuum finite elements. Mass filtering, same as in the global setting, applies to this option.

Added masses: examples



NB. Goal and method for filtering added mass is same as in the global setting

If mass filtering is activated at this level the global setup (mass filtering under Cotrol/Dynamics) is overridden.

Seismic input

The seismic input can be defined

- **in the absolute format**
  certain nodal displacements/velocities/accelerations are imposed and driven by an associated load time function
- **in the relative format**
  seismic input is given as an imposed global acceleration $\mathbf{a}_g = \ddot{\mathbf{u}}_g$ of the ground that is global to the whole structure; the corresponding inertia forces are shifted to the right hand side

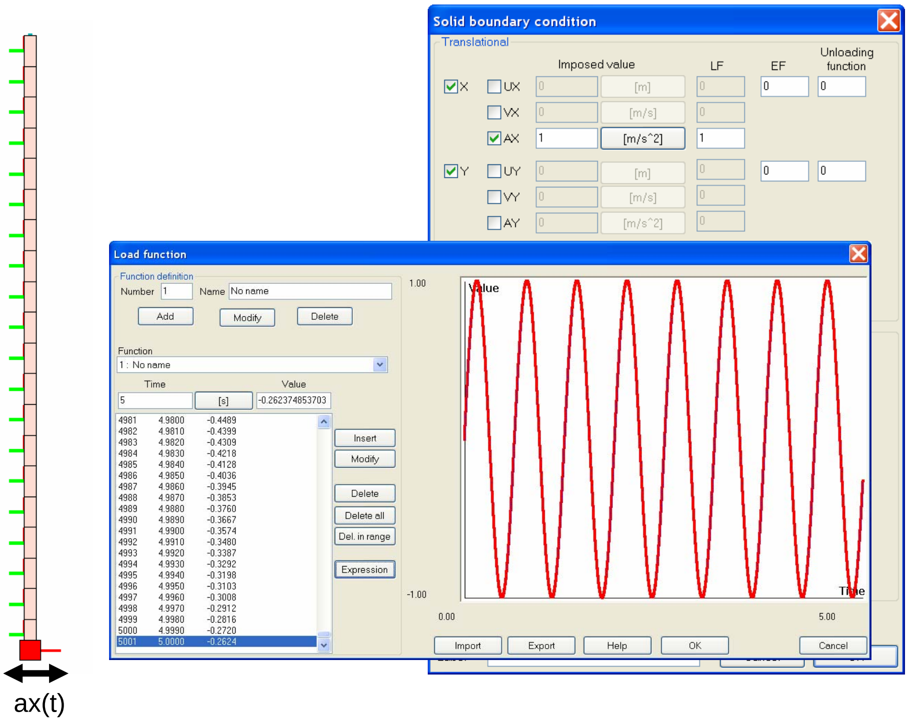
$$\mathbf{F}(t) = -\mathbf{M}\mathbf{a}_g(t) \qquad (2)$$

NB. In the absolute format displacements are total ones, referred to inertial coordinate system while in the relative one, displacements are relative ones with respect to the fixed nodes, and are referred to noninertial coordinate system (”shaking table approach”)

Note that the seismic input in the relative format can be set up exclusively in terms of accelerations while in the absolute through accelerations, velocities and displacements as well.

Seismic input: example of absolute format

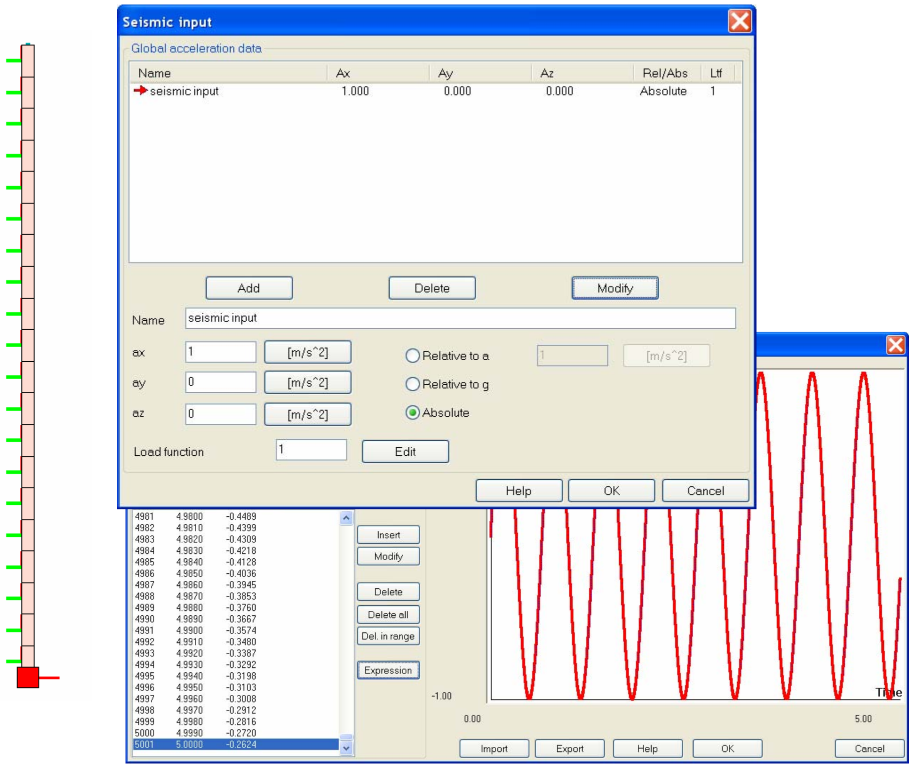
Definition through *Assembly/Preprocessing/Boundary conditions/Solid BC*



To visualize relative motion time histories one may generate standard nodal time history in the postprocessor, then export it via *.csv format and work it out in the Excel or Calc (Open Office) application. Note that 3D problems may generate huge output files while we need to store only the motion of few nodal points that are of our main concern. To do that switch to the preprocessor, make nodes selection and create labeled *Nodes group* under *Node/Create...Nodes group*. Return to the main menu, activate *Control/Results content* ☒ **Nodal time histories** option and set up the content of the output file for a given group of nodes. Results will be stored in the file *.ths that can be renamed to *.csv and worked out with the Excel or Calc (Open Office).

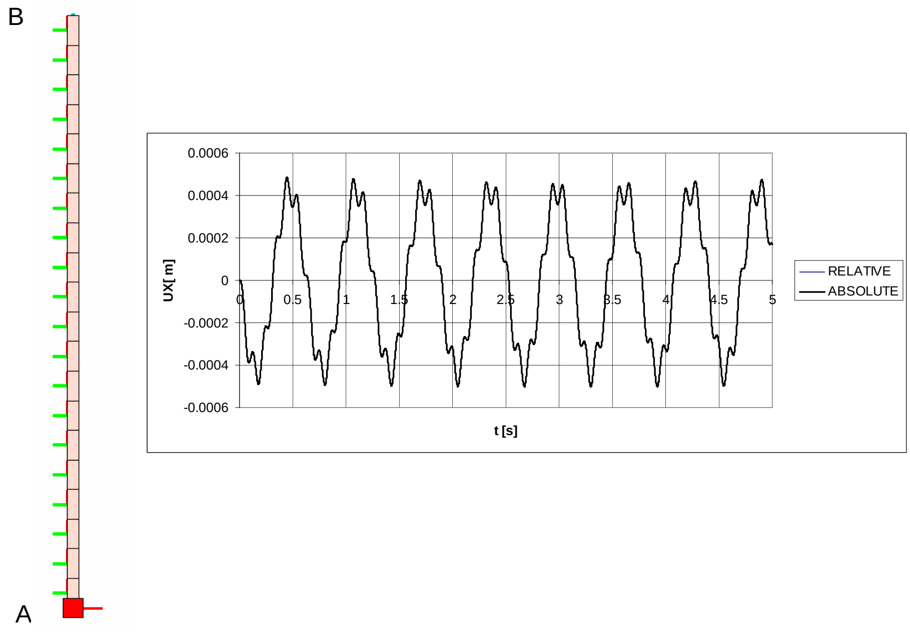
Seismic input: example of relative approach

Definition through *Assembly/Seismic input*



Here the seismic input can be set up as list of imposed acceleration time histories in a given global direction X, Y or Z. For each of the time histories a different load time function may apply and it can be set up as absolute or relative with respect to the earth acceleration g or any other value that has to be given in the dialog box.

Seismic input: comparizon of results

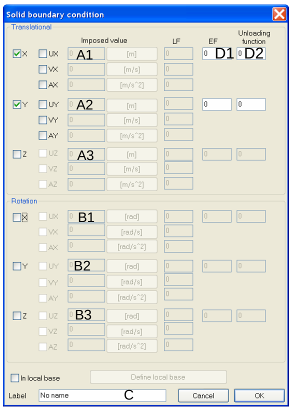


- Relative model: $u_B^{\text{rel}} = u_B$
- Absolute model: $u_B^{\text{rel}} = u_B - u_A$

Both models will yield same results. In the relative approach the rigid body motion is automatically subtracted while in the absolute one it is embedded.

Boundary conditions

Assembly/Preprocessing/Boundary conditions/Solid BC/Create: On node(s)

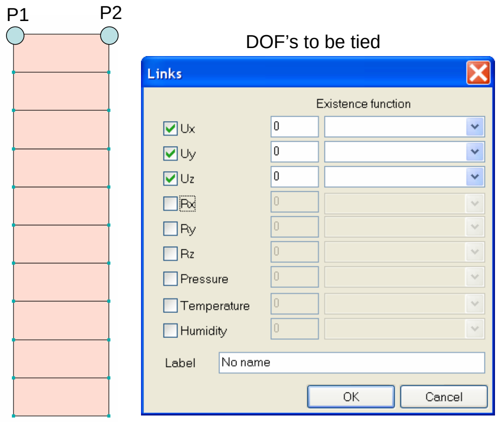


1. For dynamic driver one may impose u, $\dot{u}$ or $\ddot{u}$ at nodes
2. For dynamic driver imposed velocity or acceleration BC has a higher priority than displacement one if both displacement and velocity/acceleration BC are simultaneously active
3. To switch from rigid to viscous boundary user may define an unloading function for relaxed translational degree of freedom ; this allows to deactivate the fixities but maintain the static reactions that will preserve static equilibrium (will be discussed later one)

Compared to the static cases one may set the imposed displacements/velocities or accelerations at nodal points.
Here one may also apply an existence function and unloading function to the displacement/rotational degree of freedom. This option is very useful to run dynamic analyses preceeded by static drivers as some of the fixities (sometimes all) must be cancelled and replaced by viscous dampers.

Periodic boundary conditions

Assembly/Preprocessing/Boundary conditions/Periodic BC/Create: 2 nodes

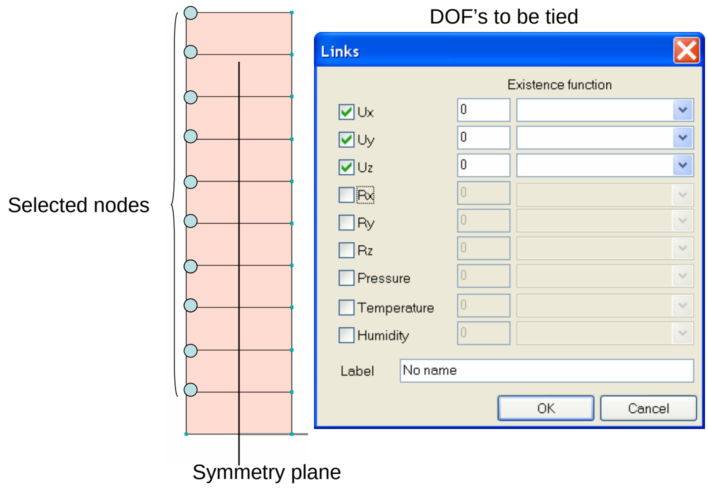


1. Tying can be made only in the global coordinate system
2. **This option does not apply to the fixed degree of freedom**
3. Each degree of freedom can be tied according to the declared existence function

Periodic BC means that we can impose conditions like $u_i^{P1} = u_i^{P2}$ and/or $p^{P1} = p^{P2}$ etc... Using this option one may easily run a shear layer problem by setting $u_x^{P1} = u_x^{P2}$ and $u_y^{P1} = u_y^{P2}$. Definition through two selected nodes can be time consuming so please refer to another possibility described in the following slide.

Periodic boundary conditions

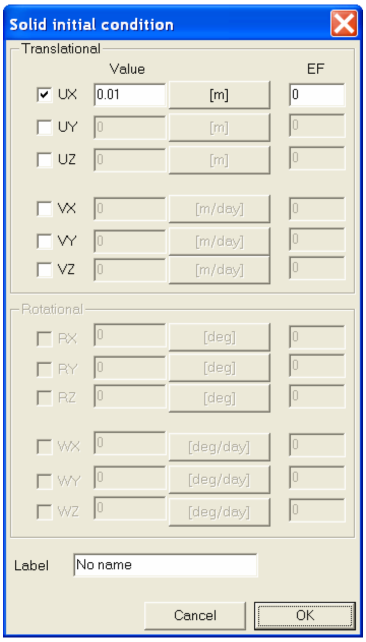
Assembly/Preprocessing/Boundary conditions/Periodic BC/Create: Nodes & Plane



- Definition through symmetry plane is useful for large computational models

To run this option you have to create an auxiliary plane and to select so called master nodes. The slave nodes will be detected automatically by clicking the symmetry plane during Periodic BC generation.

Initial conditions



1. This initial conditions can be set for displacements or velocities
2. The existence function is needed to handle the case when dynamic driver is run after some static drivers; in such case the initial displacement is understood as an increment of the displacement by which the structure will be excited

This option can be useful when making standard dynamic benchmarks or analyzing free vibrations in time domain. In the real applications the initial conditions are mostly produced by *Initial state* driver or set of static drivers following the *Initial state* but preceeding the dynamic one.

Viscous boundaries

- One of the simplest ways to avoid reflections of waves outgoing from the domain is to use Lysmer type dashpots
- The resulting damping force vector that is added to the right hand side is defined as follows

$$\mathbf{F}^{v} = -\int_{\Gamma} \mathbf{N}^{\mathsf{T}} \boldsymbol{\sigma}^{s} \mathrm{d}\Gamma$$

- $\boldsymbol{\sigma}^{s}$ is a viscous stress defined as

$$\boldsymbol{\sigma} = -\left\{ \frac{1}{c_p} (\lambda_s + 2\mu_s)\, \mathbf{n}\mathbf{n}^{\mathsf{T}} + \frac{\mu_s}{c_s} \left(\mathbf{t_1}\mathbf{t_1}^{\mathsf{T}} + \mathbf{t_2}\mathbf{t_2}^{\mathsf{T}} \right) \right\} \mathbf{v}^{\mathbf{s}}$$

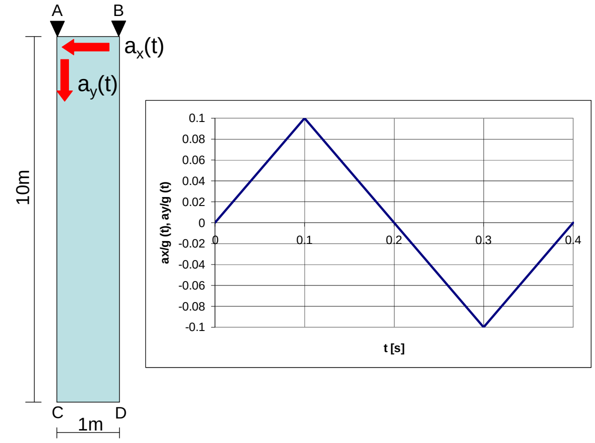
- The corresponding shear and dilatational wave velocities are denoted by

$$c_s = \sqrt{\frac{G}{\rho}} \qquad c_p = \sqrt{\frac{\lambda + 2\,G}{\rho}}$$

NB. The normalized normal and tangential vectors are denoted by $\mathbf{n}$ and $\mathbf{t}_1$, $\mathbf{t}_2$

Note that Lysmer type viscous dampers are the simplest elements that help to cancel wave reflections from domain boundaries. However only in some certain situations these elements give results with high accuracy or even exact. This strongly depends on the angle of wave incidence.

Viscous boundaries: example



- Viscous damper is added at C-D boundary

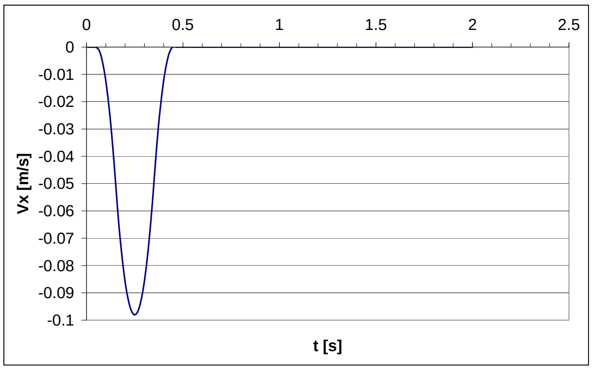
- Material data:
- $E = 200000$
- $\nu = 0.25$
- $\gamma = 9.81$ [kN/m^3]
- $\rho = \dfrac{\gamma}{g} = 1000$ [kg/m^3]
- $c_s = \sqrt{\dfrac{G}{\rho}} = 200$ m/s
- $c_p = \sqrt{\dfrac{\lambda + 2\,G}{\rho}} = 346.41$ m/s

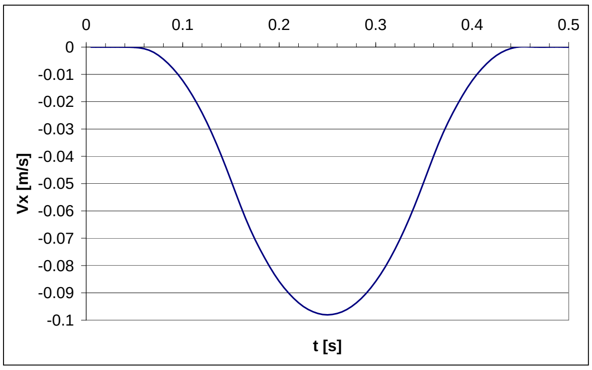
NB. Viscous dampers may inherit their properties from adjacent continuum elements (to be set at material level)

The aim of this test is to show that after certain time the signal sent from boundary A-B hits the boundary C-D and is not reflected. The signal is imposed at nodes A and B through imposed accelerations a_x and a_y according to the load time function as shown in the figure.

Viscous boundaries: example

- horizontal velocity at points C,D should vanish after time $t = 0.4 + \frac{10}{200} = 0.45$ s

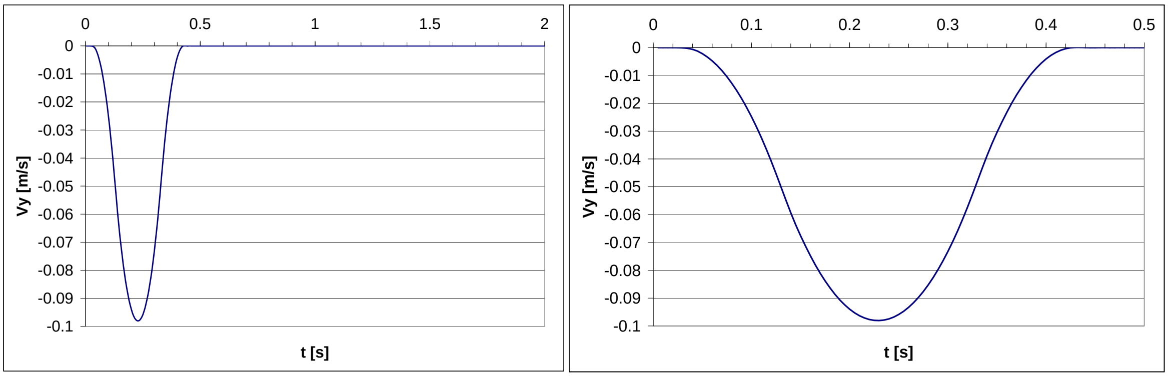




The left plot shows horizontal velocity time history in the period of 5s. The right plot is just a time zoom from $t = 0..0.5$ s. Note that the exact and numerical solution match very well.

Viscous boundaries: example

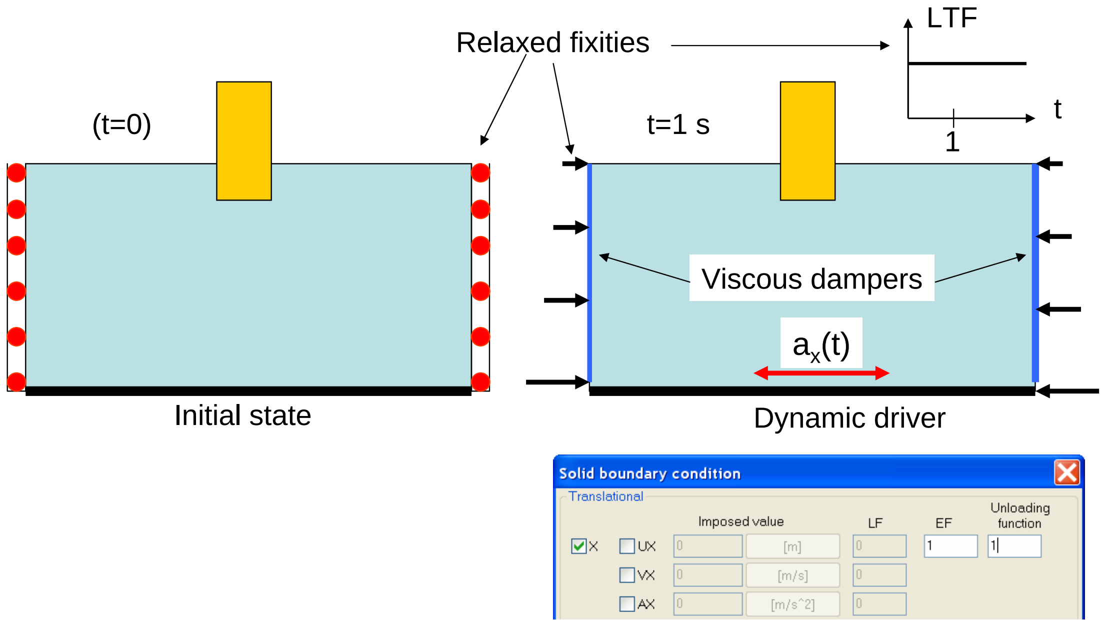
- vertical velocity should vanish after time
  $$t = 0.4 + \frac{10}{346.41} = 0.429 \text{ s}$$



The left plot shows vertical velocity time history in the period of 5s. The right plot is just a time zoom from $t = 0..0.5$ s. Note that the exact and numerical solution also match very well.

Viscous boundaries in practise

How to handle transient dynamics preceeded by an initial state or other static drivers ?



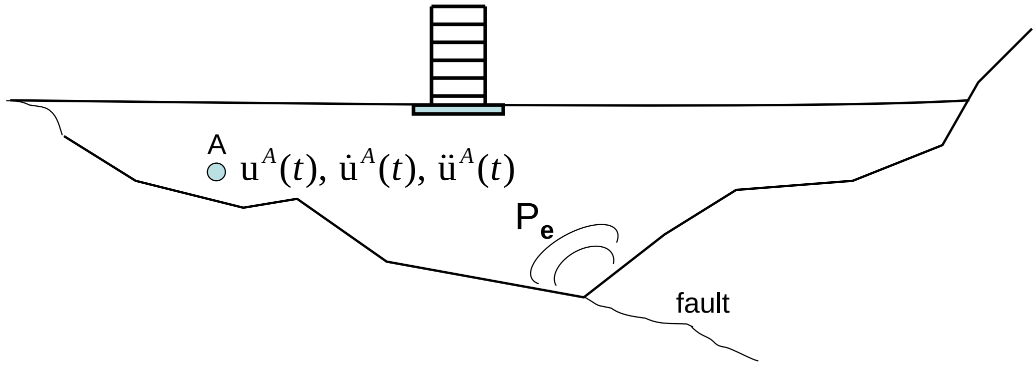
In the practical cases we start computation from the initial state driver that can be followed later by static drivers (to model excavation/stage construction). At this stage we have to keep rollers along vertical walls of the model. Once we switch to dynamic driver (assuming that the bottom part is just a bedrock) these rollers must be cancelled and replaced by viscous dampers. This operation can be made by setting the existence function for these rollers and the unloading function that is equal to 1.0 (no dissipation of horizontal reactions) all the time. This way we keep static reactions that preserve static equilibrium while the boundary becomes unsupported. The viscous dampers must be activated at the beggining of the dynamic driver (via existence function).

Domain Reduction Method (DRM)

- The main goal: analyze computational model that concerns the structure and only a small adjacent part of subsoil
- Size of the problem to be solved is substantially reduced
- The Domain Reduction Method (DRM) was proposed by J. Bielak et al. (2001)

Using this method one may solve problems (mainly in 3D and mostly including continuum elements) that you would not be able to solve using implicit methods on PC platforms (even multicore and 64 bit). Models of size 400 000 DOFs (under 64 bit systems with 8 Gb of RAM) can be run within 6 to 12 hours. To solve larger models in dynamics dedicated 64 bit PC with 32 Gb RAM (or more) are recommended.

DRM: general idea

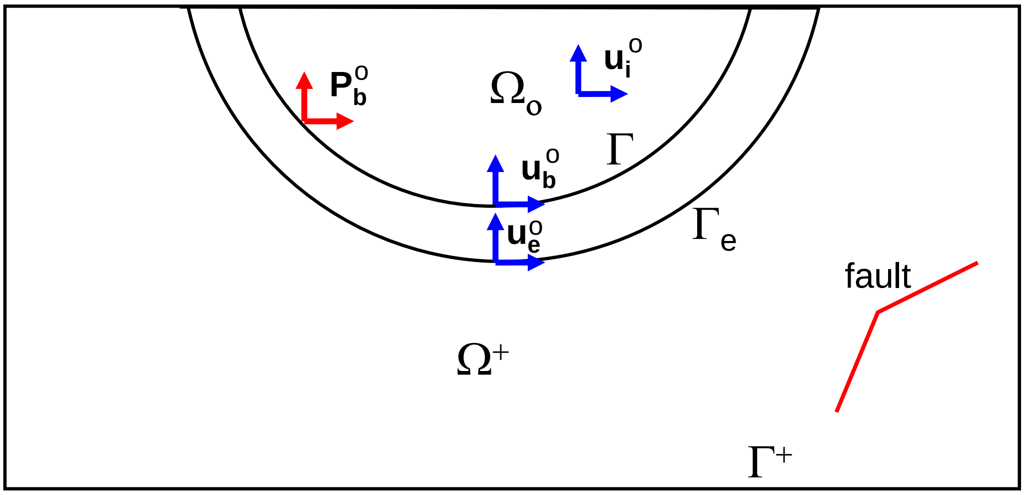


Full model of subsoil and structure, and source of the loading $\mathbf{P}_e(t)$

1. At any point displacements, velocities and accelerations induced by $\mathbf{P}_e(t)$ are denoted by $\mathbf{u}(t)$, $\dot{\mathbf{u}}(t)$, $\ddot{\mathbf{u}}(t)$
2. This model with a large subsoil zone and source of load $\mathbf{P}_e(t)$ is decomposed into two models:
   - background model
   - reduced model

Here we want to decompose te analysis on two steps. In the first one we analyze the free field motion (without structure) while in the second motion of only part of the domain of the interest including the structure. It is possible to analyze the free field in 2D or axisymmetric format while the near field is solved as 3D.

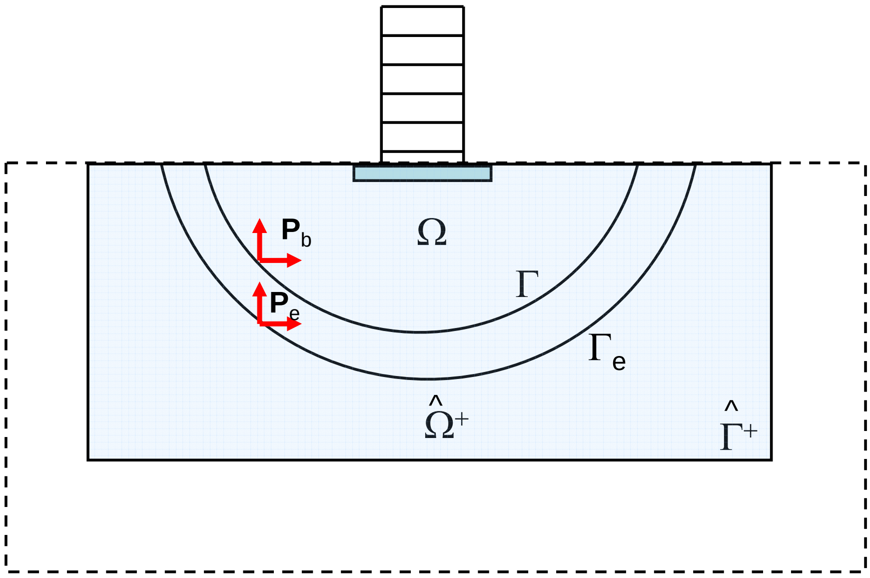
DRM: Background model



1. In the background model the structure is removed and free field motion is analyzed
2. Displacements, velocities and accelerations induced by $\mathbf{P}_e(t)$ are denoted by $\mathbf{u}^0(t)$, $\dot{\mathbf{u}}^0(t)$, $\ddot{\mathbf{u}}^0(t)$

At this stage we store displacements, velocities, accelerations at nodes that belong to the boundary layer of elements. These quantities are used later on to compute effective forces that are applied through the boundary between exterior and interior domain. The detailed derivation of the approach can be found in the dedicated report on dynamics in Z_Soil.

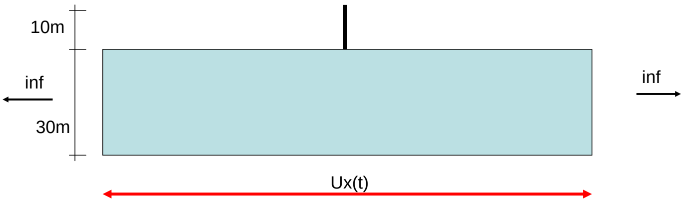
DRM: Reduced model

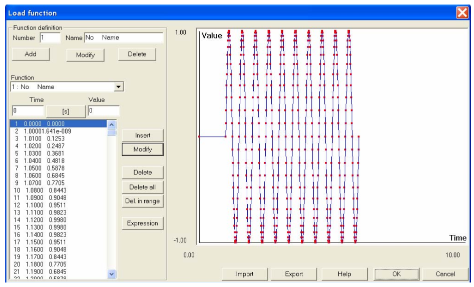


1. $\hat{\Gamma}^+$ is a boundary where viscous damping elements are to be put to cancel wave reflections
2. Displacement decomposition in the exterior domain: $\mathbf{u}_e = \mathbf{u}_e^0 + \mathbf{w}_e$

In the exterior domain we seek for the residual motion with respect to the free field. Hence if we had taken a free field that includes the structure the residual motion would vanish to zero.

DRM example: 2D soil-column interaction

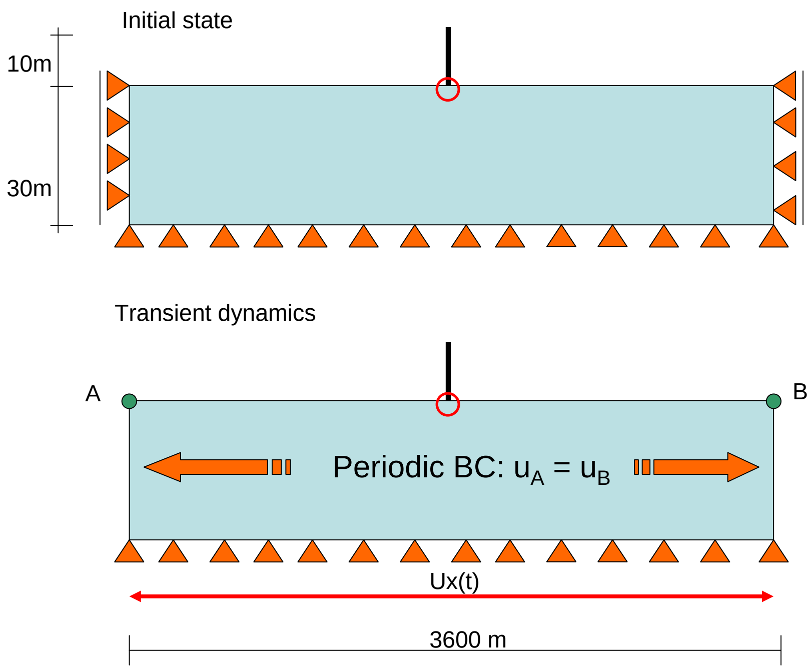




- Continuum material data: $E = 192000$ kPa, $\nu = 0.2$, $\rho = 2000$ kg/m^3
- Beam material data: $E = 20000000$ kPa, $\nu = 0.2$, $\rho = 2500$ kg/m^3, $A = 1\text{m}^2$
- Imposed base displacements: $u_x(t) = sin(2\,\pi\,(t-1)) \times 1m$ in time range $t = 1..7$s
- Periodic BC are enforced for both walls of the layer (for all displacement components)
- Time stepping: $\Delta t = 0.01$s
- Time duration: $t = 1..6$s
- Element size: $h^e = 2$m

In this example we solve first a huge model that is 3.6 km long by applying the excitation through the imposed displacements at the base assuming the periodic BC at both vertical walls of the model. By setting this periodic BC, in case of lack of the structure, we would obtain a motion that is compatible with the shear layer solution. However, the structure interacts with the soil and the resulting motion is not anymore compatible with the shear layer mode. Solution of this large model we will treat as a reference one to assess the accuracy of the DRM method.

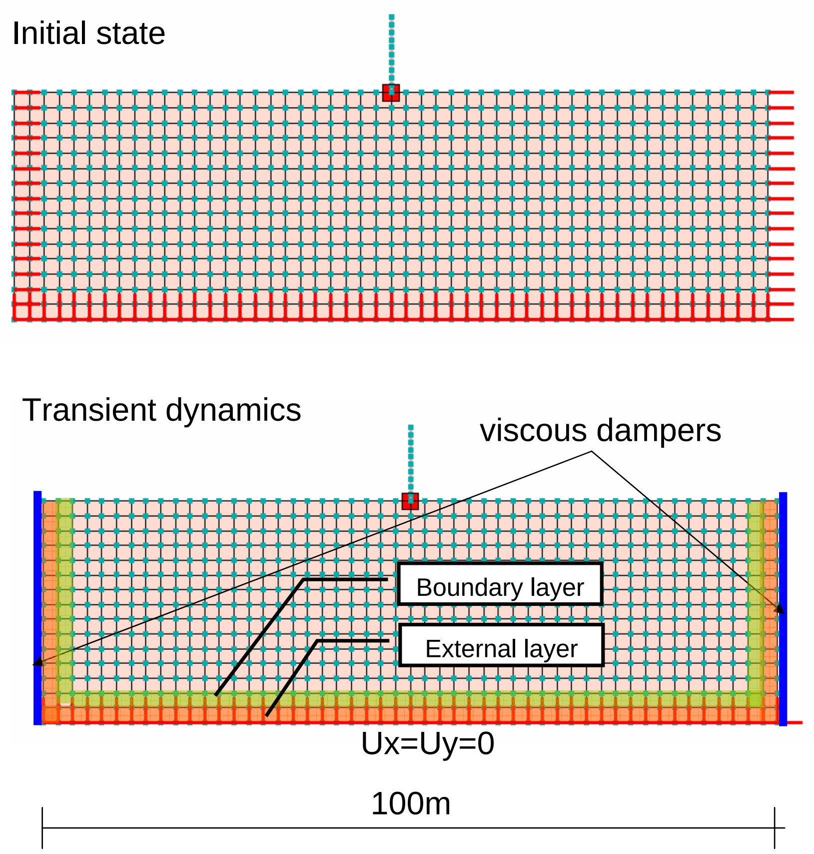
DRM example: Full model



- 27000 Q4 elements are used + 10 beam elements

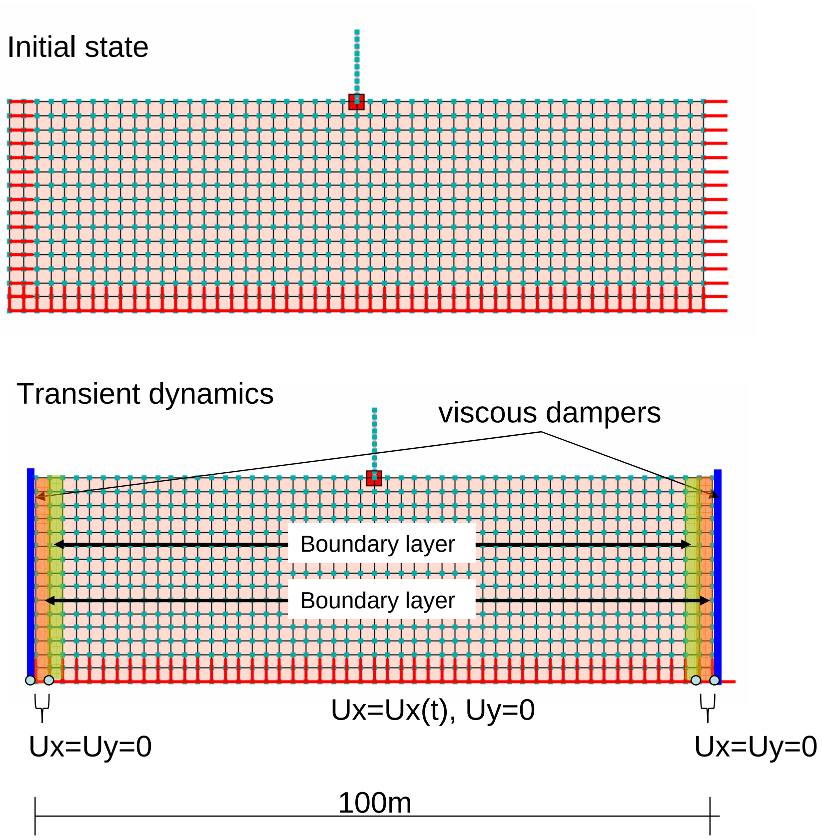
In the initial state we keep the vertical rollers. Once the dynamic driver is activated these vertical rollers are cancelled with the unloading function equal to 1.0 all the time. Simulatenously periodic BC are activated. Note that in this example we do not need to activate viscous dampers as we solve the problem till time when the waves hit the vertical walls of the model.

DRM example: DRM model 1



The DRM model is 100 m long and its depth extends up to the base. The first possible DRM setup is such that the first external layer of elements is defined as the exterior domain while the second one is the boundary layer. The DRM model is solved as a sequence of the initial state driver with vertical rollers active followed then by the dynamic driver in which rollers are cancelled, unloading (with LTF=1.0) takes place, viscous dashpots are placed on both vertical walls and we fix the nodes at the base with the imposed displacements equal to 0.0. This is so because we seek in the exterior domain for the residual field and base nodes must have a residual field equal exactly to zero.

DRM example: DRM model 2



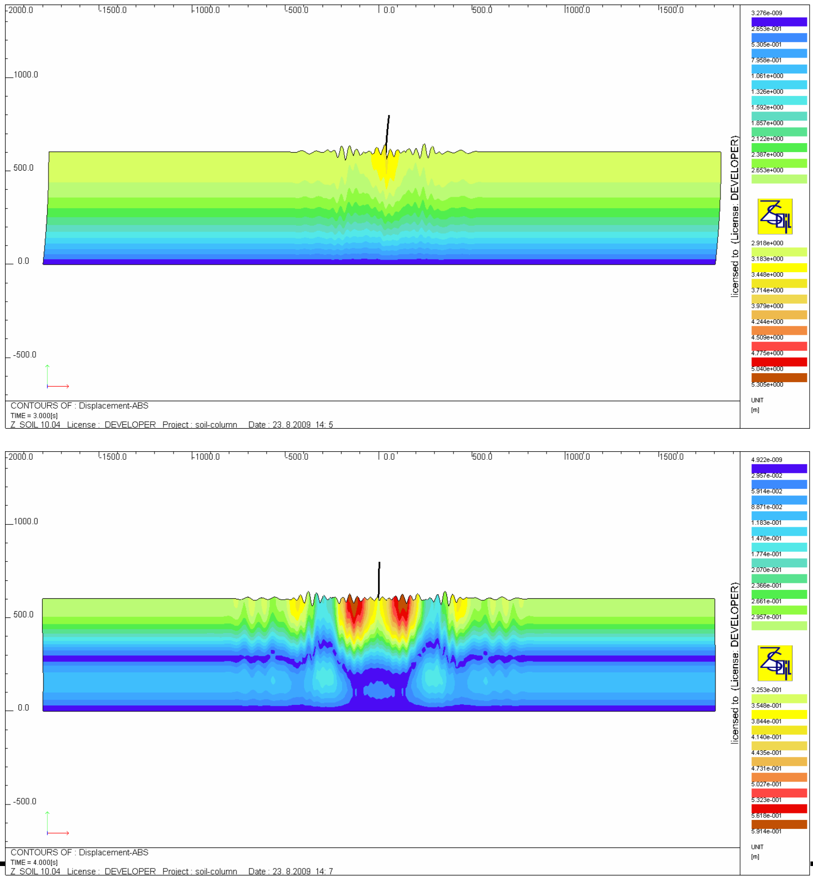
The second DRM model is similar to the first one but exterior and boundary layers are defined exclusively along vertical walls. The major difference between the two DRM models is such that at the base nodes, that are in the interior domain, we must apply the same excitation as in the full model, while in the exterior nodes we impose zero displacements. So we see that this setup is somewhat more complicated.

DRM example: sequence of analyzes

- Solve full model subsoil-beam (FULL) $\Longrightarrow$ reference solution (3600m long, 1800$\times$ 15=27000 Q4 elements+10 beam elements)
- Solve simple model (SHL) $\Longrightarrow$ free field motion (3600m long, 1$\times$ 15=15 Q4 elements)
- Solve DRM-1 model (DRM-1-FF-FULL) with free field motion from full model
- Solve DRM-1 model (DRM-1-FF-SHL) with free field motion from simple model
- Solve DRM-2 model (DRM-2-FF-FULL) with free field motion from full model
- Solve DRM-2 model (DRM-2-FF-SHL) with free field motion from simple model

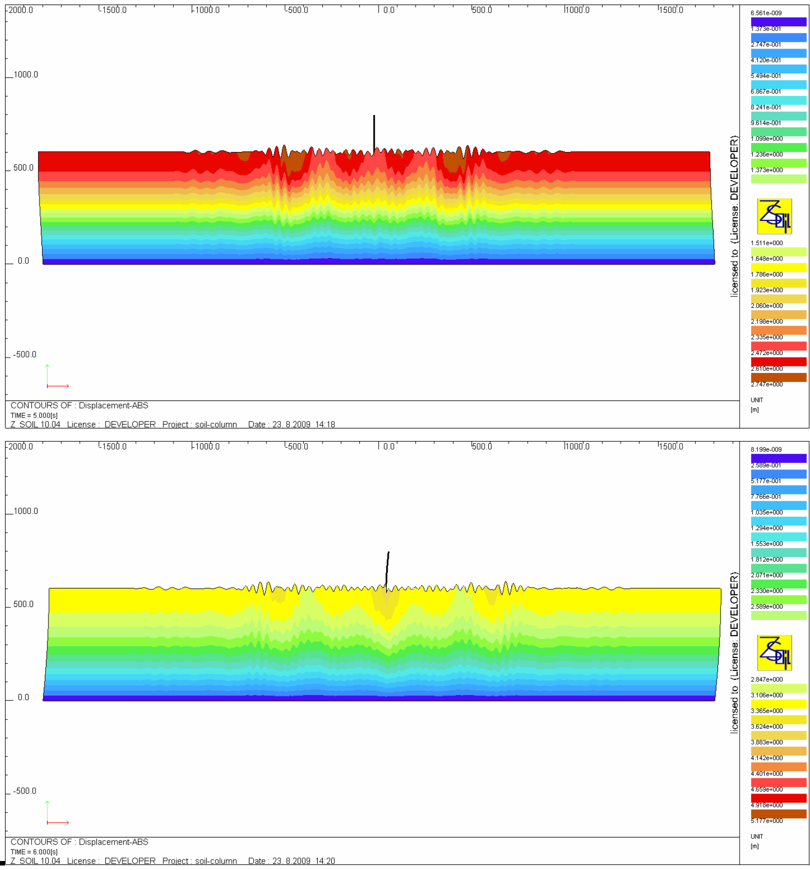
To verify the accuracy of the method we solve first the huge 3.6 km long model as a reference one. Then we solve an auxiliary 1D model (although via 2D elements and periodic BC) that represents a motion of shear layer and we will call it as a simple model. Then we run the DRM-1 model once with the free field taken from full model and once from simple model. We do the same for the DRM-2 model.

Results for full model: Deformation after 2s, 3s of excitation



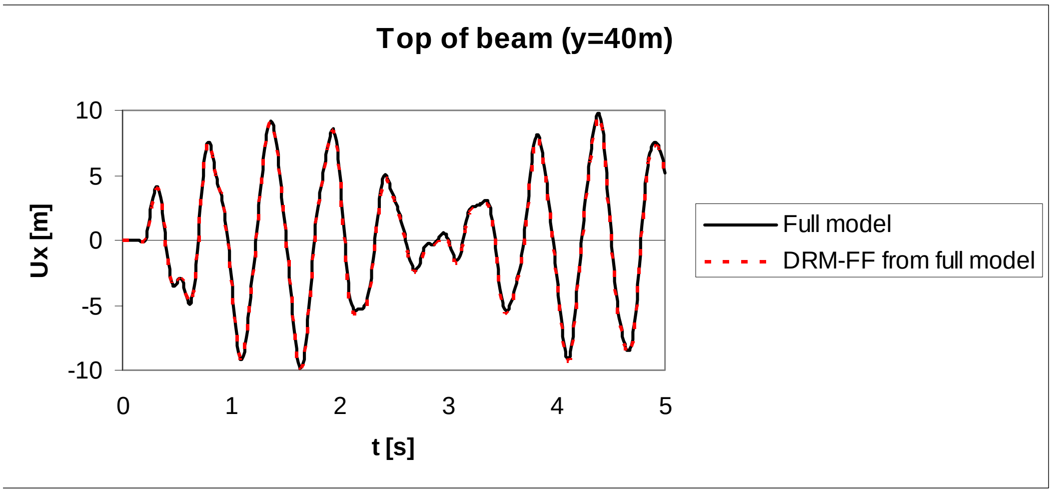
These plots show the colour contours of displacement field for huge model after 2s and 3s respectively. We can see that the signal is far from the boundaries. The motion which is well visible corresponds to the Rayleigh waves.

Results for full model: deformation after 4s, 5s of excitation



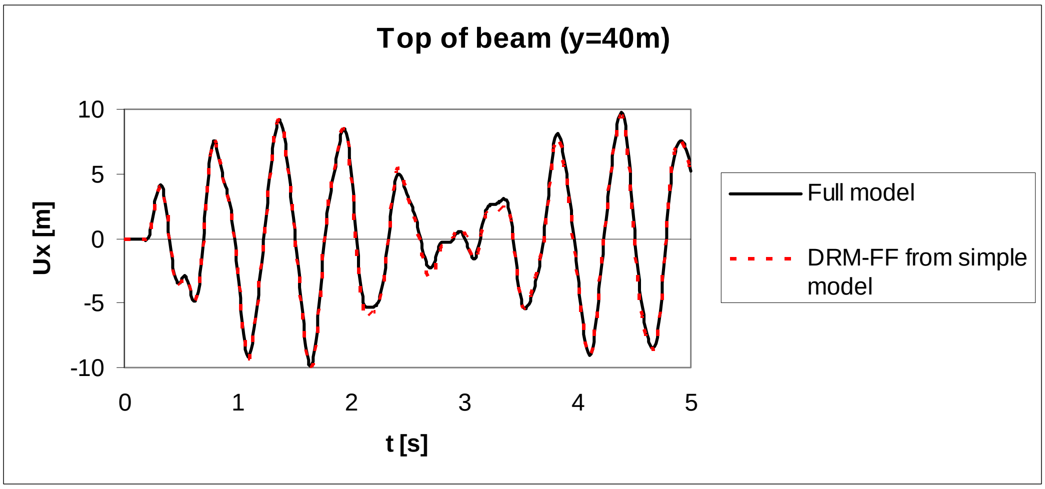
These plots show the colour contours of displacement field for huge model after 4s and 5s respectively. We can see that the signal is still at some distance from the boundaries. The motion which is well visible corresponds to the Rayleigh waves.

Results for DRM model-1 with free field from full model



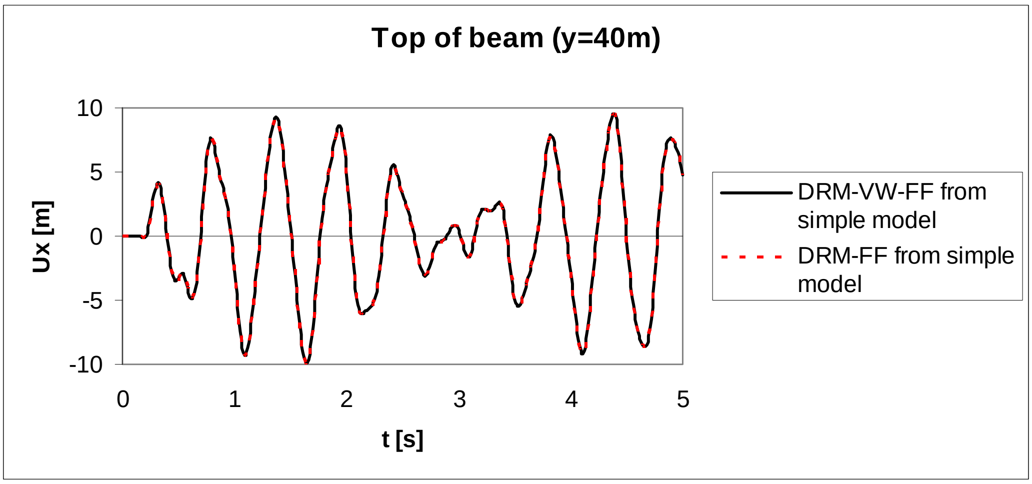
This plot shows displacement time histories $u_x(t)$ of the tip of the beam generated by the DRM-1 model with the free field taken from full model and by full model. These results must be exactly the same that proves that the implementation of the method is correct.

Results for DRM model-1 with free field from shear layer model



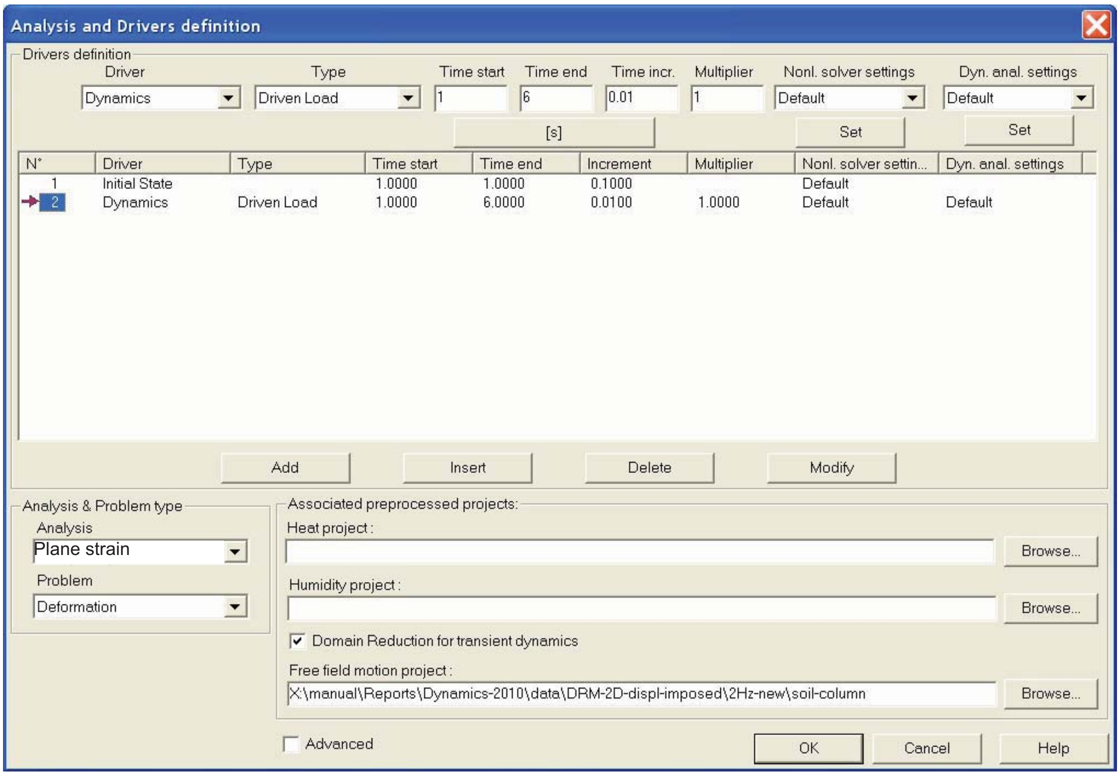
This plot shows displacement time histories $u_x(t)$ of the tip of the beam generated by the DRM-1 model with the free field taken from simple model and by full model. These results are practically the same and little deviation comes from the fact that the periodicity condition is satisfied only at the vertical walls and not in the nearest neighbourhood of these walls. Note that the DRM model is 30 time smaller than the huge one and with the DRM model one can continue the analysis after time $t = 5$ s that is when the signal hits the vertical boundaries.

Comparizon of DRM model-1 and DRM model-2 (both with free field from shear layer model)



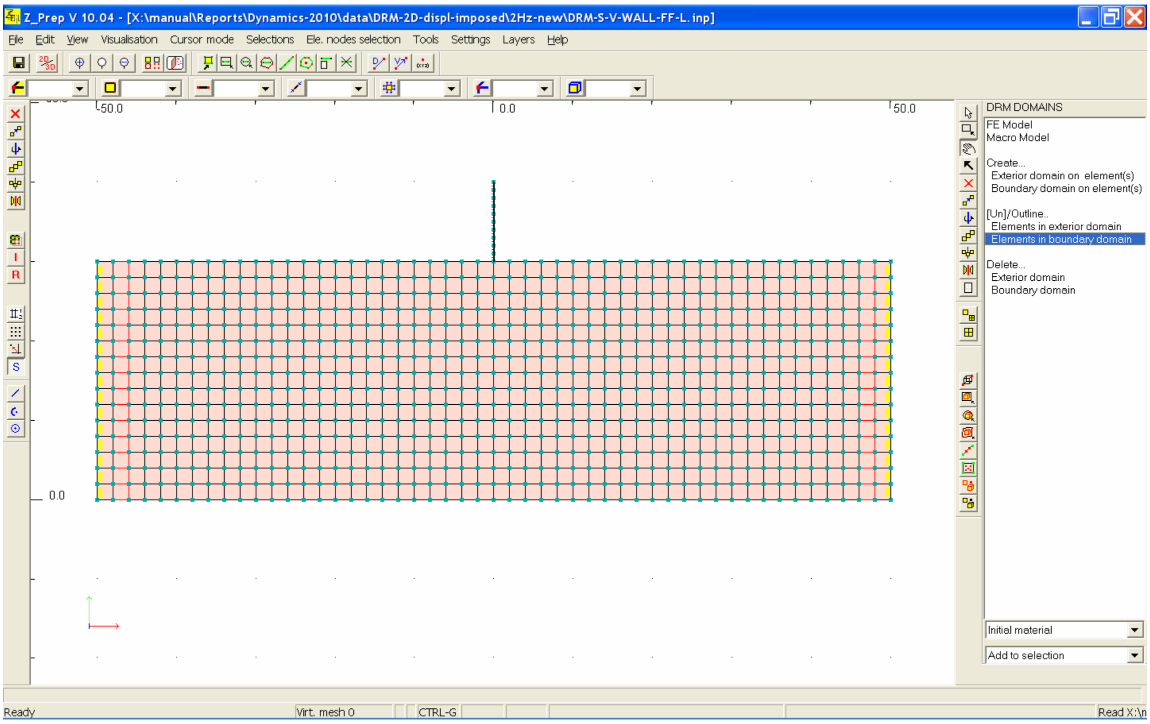
This plot shows displacement time histories $u_x(t)$ of the tip of the beam generated by the DRM-1 and DRM-2 models computed with the free field taken from simple model. These results are exactly the same.

DRM user interface: adding free field solution for tansient dynamic driver



To apply the free field motion project we have to point on the dynamic driver in the list of drivers first and then browse for it with standard Windows file manager. Note that the full path to the free field is stored so the data transfered from one computer to another may not directly work if the DRM model and free field are kept in different directories.

DRM user interface: setting exterior/boundary domain



Setting of the exterior and boundary layers can be made at finite element level during preprocessing stage. Use option *FE model/DRM domains/Create...*