4. Modeling of soil structure interaction

Dynamic soil-structure interaction (DSSI) occurs when a structure and the soil on which it is founded interact during a dynamic event such as an earthquake. There are different methods to take this interaction into account. The most common are the direct, the substructure and the hybrid methods. At Arup, as practitioners and being LS-DYNA a finite element analysis program, we normally use the direct approach, the substructure approach, or a combination of both to solve DSSI problems. The choice between one or the other strategy will depend on many factors: type of project, seismicity of the area, type of structure, …

Direct approach

The most computationally expensive alternative, where the soil and the structure are both modeled using finite elements. It's typically only undertaken on large, critical projects like nuclear power plants or large infrastructure projects such as major bridges, tunnels, subway stations, tanks and marine structures, and requires specialist expertise. It's able to easily incorporate all the material and geometric nonlinearities in the soil and soil-foundation interface. In some cases, the effort in modeling could be justified for the sake of designing more cost-effective structures, enhancing resilience, and adding value to our clients.

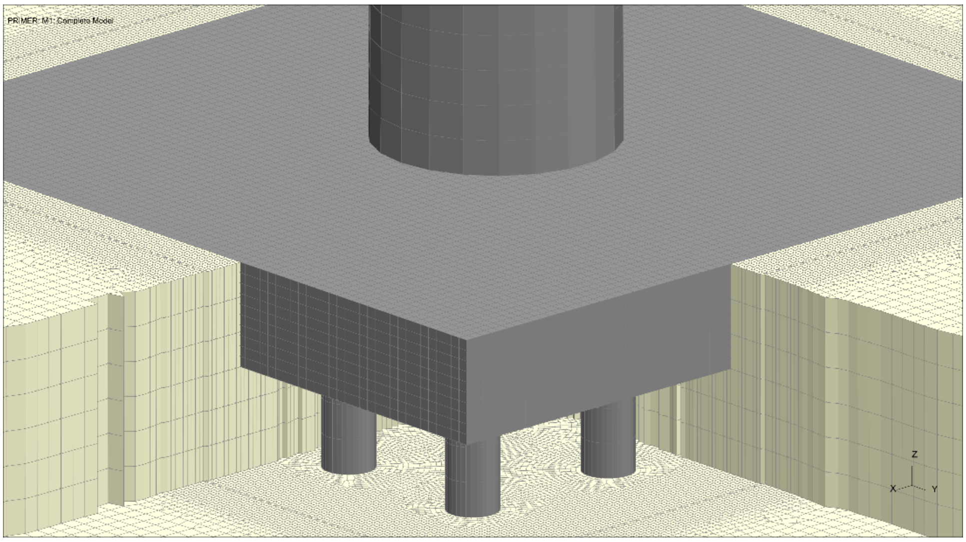


Figure 1 Direct approach example using a global finite element model.

Substructure approach

The substructure method decomposes the complete soil– foundation–structure system into several subdomains and, underlain by the assumption of geometrical and material linearity, evaluates the response of each subdomain separately and superposes to obtain the total system response. This technique presents some appealing advantages over the direct SSI approach, such as the physical insight (inertial al kinematic interaction thinking) and the low computational effort. However, requires the definition of several steps (i.e. creating several models) which may not be easy to carry out or include many simplifications. The typical steps are:

- Obtention of a foundation input motion (FIM). This might include running a Site Response Analysis and extract surface ground motions.
- Definition of the dynamic impedance functions – springs and dashpots – to represent the frequency-dependent stiffness and damping characteristics of soil-foundation interaction.
- Final model with the superstructure modeled above the foundation and the dynamic impedance functions and excited with the FIM.

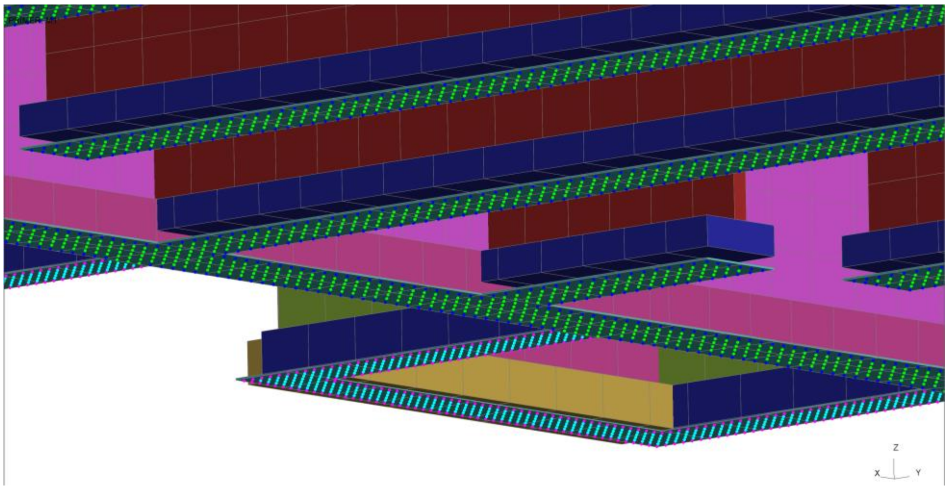


Figure 1 URM building with strip footings mounted on soil springs

Modified direct approach

This modeling strategy has been used extensively in the P500 project. It combines the simplified modelling techniques commonly used for substructure techniques (dynamic impedance functions) with the direct approach, i.e. combine the sub-models to create a single LS-DYNA model. This strategy may be preferred to that of a substructure because in one single model you can incorporate all the relevant DSSI effects in the analysis, as in the direct approach. It is very useful when piles are presented, because applying foundation input motions (FIM) along the length of the pile can be a laborious task, as the input motions are likely to vary with depth.

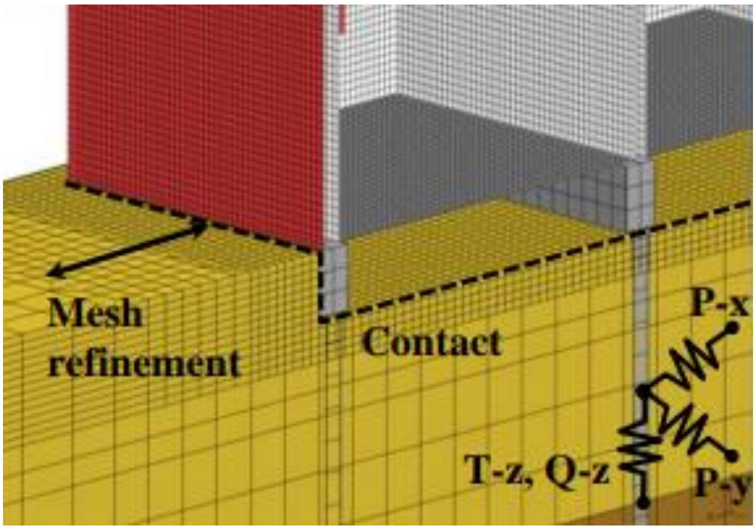


Figure 1 Modified direct approach mixing FE and soil springs.

The following sections define how the elements involved in the DSSI can be modelled: the soil domain (4.1), the foundation (4.2) and the interfaces between them (4.3). There is also a 4.4 section with some watch-its to be aware of.

4.1 Modeling of the soil domain

The modeling of the soil domain is quite complex in a dynamic SSI problem. The soil domain has to be able to:

- Reproduce the dynamic properties of the soil.
- Capture the nonlinearities and failure modes occurring during the analysis (near field vs far field).
- Satisfy the radiation condition at the boundaries towards infinity.

The soil domain is discretized into layers which represent soil with different properties. Solid elements are frequently used. The thickness of each soil layer needs to be carefully selected to ensure 1) ground motion frequencies can pass through it and 2) "numerical noise" is not present. The maximum passable frequency of a soil layer is defined as:

$$f_{max} = \frac{V_s}{4H}\ [Hz]$$

Thus, a very thick layer will have a small maximum frequency and some frequencies risk to be "stuck" on it. On the other side, with a high f_{max} (i.e. very thin layer), high frequency "numerical noise" could appear, leading to a subsequent filtering to correct the computed data. Based on past projects, at Arup we normally recommend 25Hz < f_{max} < 50Hz. However, the maximum layer frequency can be ignored in favor of greater control over the dynamic soil properties with depth.

4.1.1 Material model

LS-DYNA has an extensive material library that can be used to model the soil. Some of them are:

- * MAT_SOIL_AND_FOAM (*MAT_005).
- *MAT_HYSTERETIC_SOIL (*MAT_079).
- *MAT_MOHR_COULOMB (*MAT_173).
- *MAT_SOIL_BRICK (*MAT_192).
- *MAT_DRUCKER_PRAGER (*MAT_193).
- *MAT_SOIL_SANISAND (developmental).

At Arup, of all of them, the most widely used for seismic applications is *MAT_HYSTERETIC_SOIL (*MAT_079). It is a nested bounding surface plasticity model, well suited to reproduce the hysteretic, nonlinear behavior of soils under cyclic excitation. At a minimum, it requires:

- RO: density
- K0: bulk modulus

- LCID: backbone curve of shear strain versus shear stress
- (Optional but recommended) LCD: curve of shear strain versus damping ratio. If 0, the model exhibits Masing damping.

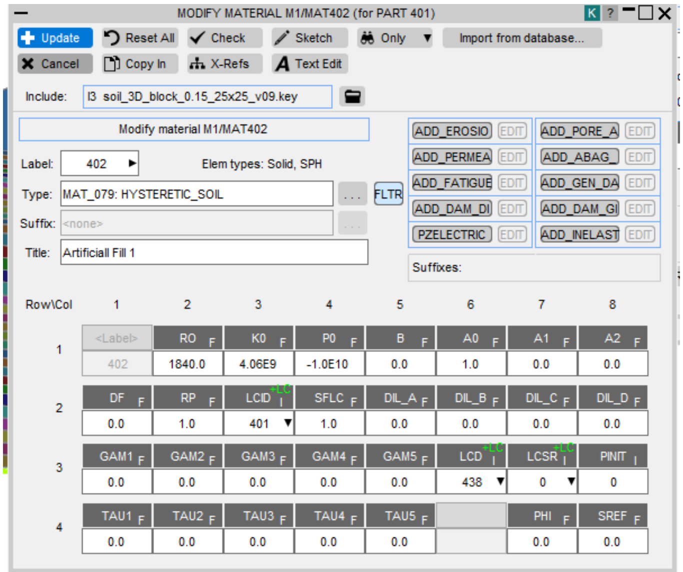


Figure 1 Material card used for Material 079

One of the main limitations of this material model is that it is not able to generate pore pressure. There are some parameters in the material card (DIL_A through DIL_D) which try to introduce a dilatant behavior but are too simplistic. Therefore, all the analyses carried out with this material model are total stress analyses (TSA), i.e. undrained conditions.

For more information on *MAT_HYSTERETIC_SOIL read the document: https://arup.sharepoint.com/sites/seismic/SiteCollectionDocuments/Forms/Display%20View.aspx?id=%2Fsites%2Fseismic%2FSiteCollectionDocuments%2FLearning%2FAmericas%2FMAT79%5Ftraining%5FBC%5FHydro%5FFinal%5FforIssue%2Epdf&parent=%2Fsites%2Fseismic%2FSiteCollectionDocuments%2FLearning%2FAmericas

4.1.2 Mesh refinement

Depending on the approach to solve the DSSI problem, we are more demanding on the finite element mesh. For example, in the direct approach the mesh must be able to, besides capturing non-linear site-response and kinematic interaction between foundation and soil, capture all soil-structure interaction phenomena that occur between the foundation elements and the soil. This is only possible with a very refined mesh. However, if we use dynamic impedance functions (springs and dashpots) to capture the inertial interaction, to only reproduce the non-linear site-response and kinematic interaction between foundation and soil we can have a soil domain coarsely meshed.

We will concentrate on the first case, where we need a very refined mesh in the model. One of the main drawbacks of the direct approach is its high computational cost. To help overcoming this limitation, it is proposed to divide the supporting soil domain into two sub-domains:

- a *far field domain*, which extends a sufficient distance from the foundation for the soil structure interaction non linearities to be negligible; non linearities in this domain are only governed by the propagation of the seismic waves.

- a *near field domain*, in the vicinity of the foundation where all the geometrical and material non linearities due to soil structure interaction are concentrated.

The idea is to have a concentrated, computationally expensive area around the structure where the mesh is refined enough to explicitly capture local soil-foundation flexibility, bearing failure, soil yielding and geometrical nonlinearities while having a coarse, less computationally expensive mesh until reach the lateral boundaries.

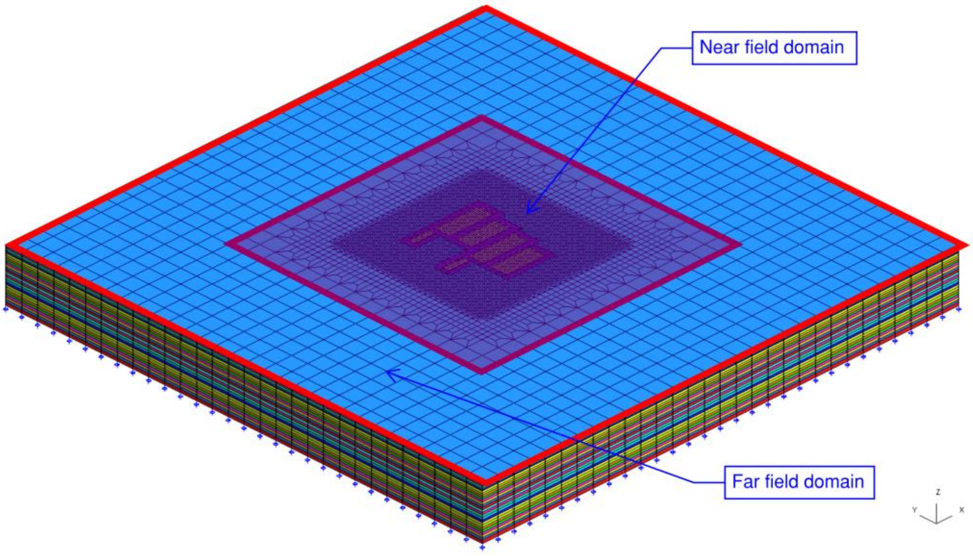


Figure 1 Schematic definition of the far and near field domains.

A key factor to take care of in this modeling strategy is the mesh transition between the near field and the far field domain. It is advisable to do it both vertically and horizontally, to optimize the computation time to the maximum. A common approach is to use a mesher to define the horizontal transition and a contact in LS-DYNA in the vertical direction.

4.1.2.1 Horizontal transition

This mesh transition is done at a certain distance from the foundation and can be defined with any mesher available.

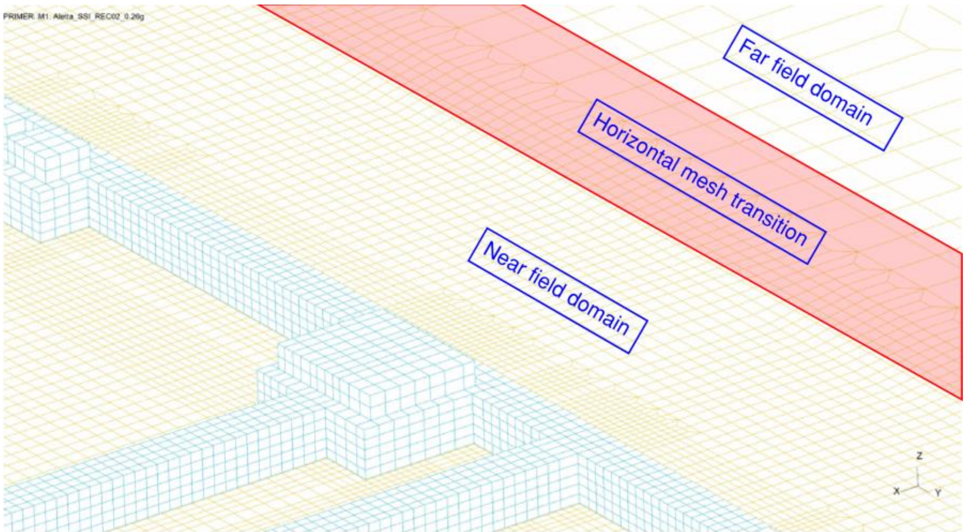


Figure 1 Example of a very structured mesh transition.

A first option is the Oasys software GSA. 2D members can be defined in the design layer with <u>different element sizes</u> to create the mesh transition from the near field to the far field.

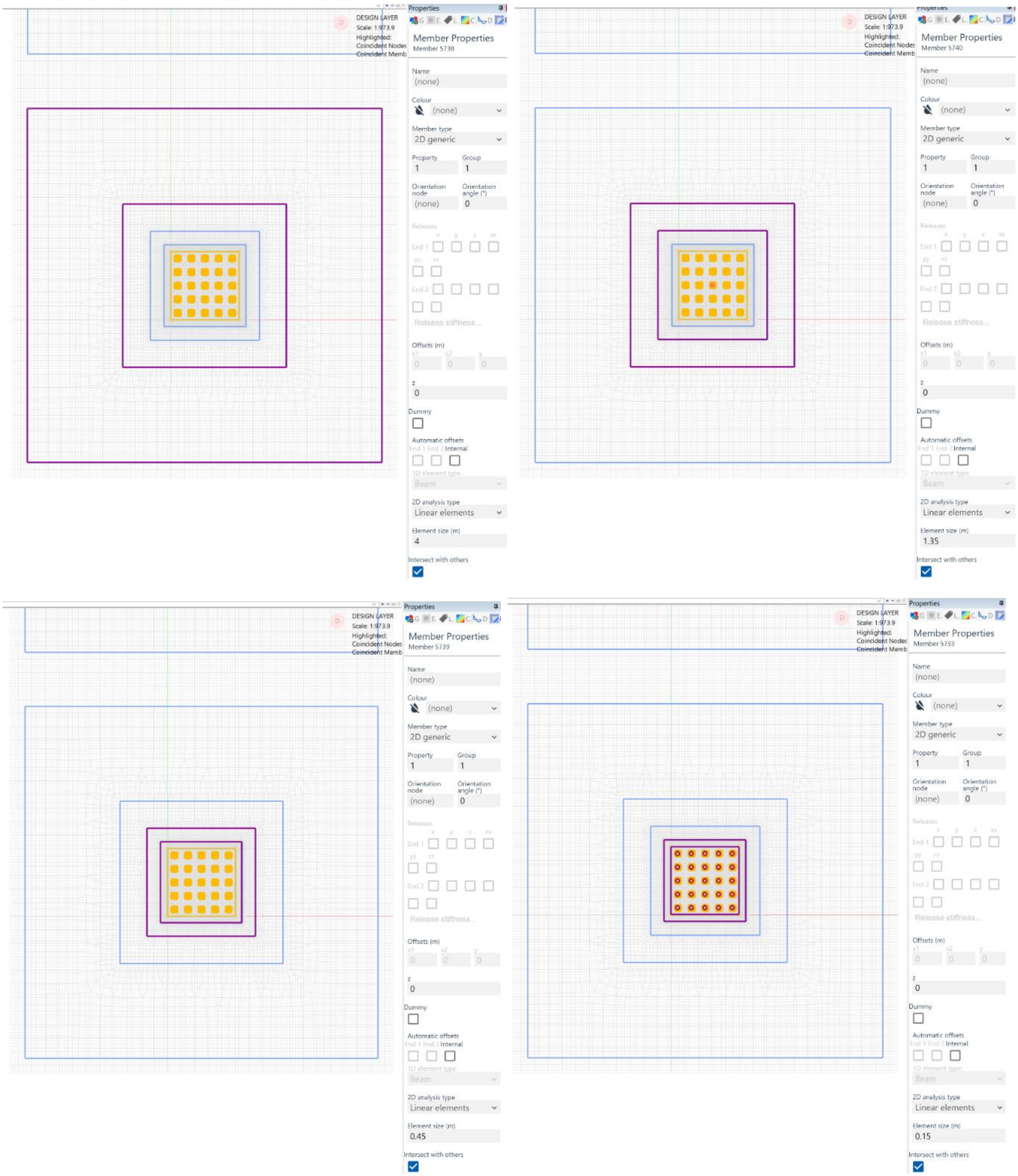


Figure 1 Members with different element sizes to create the mesh transition.

Moving elements from the design layer to the analysis layer allows, by using the Analysis Wizard, creating a .key file.

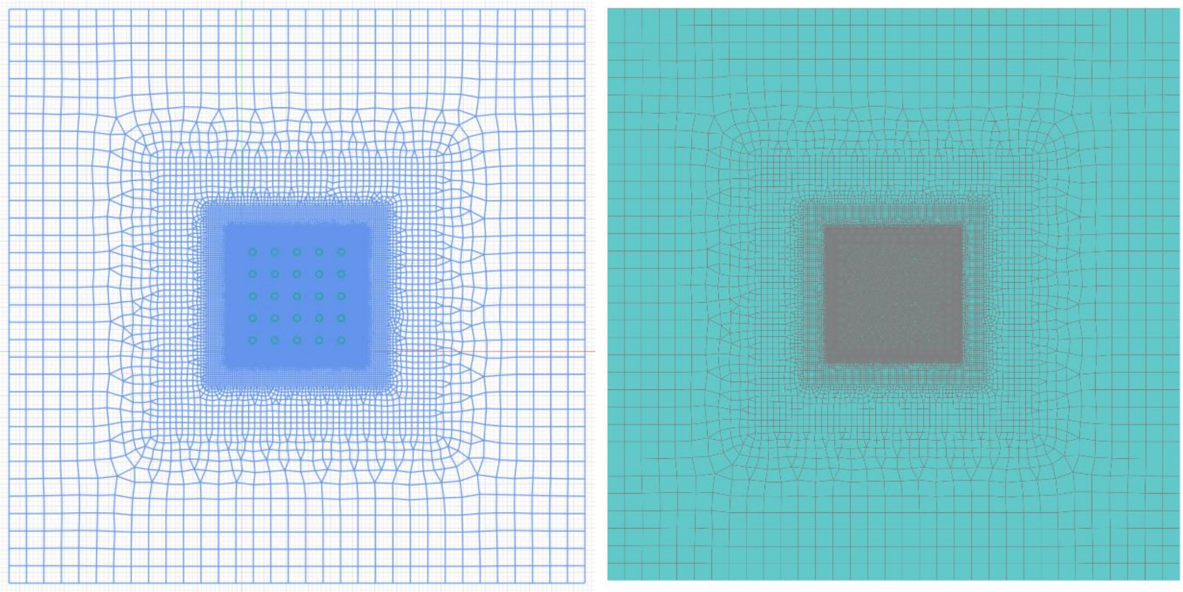


Figure 1 GSA mesh (left) and the corresponding .key file (right) with the mesh transition.

Another option is to use the *20201007_Soil_Block_Surface_Mesher_v05.8 UI.gh* script (https://github.com/arup-group/gesuautomation-structuralautomation/blob/master/1_2%20Meshing_Script/Soil%20Meshing/20201007_Soil_Block_Surface_Mesher_v05.8%20UI.gh) which uses a Griddle license to mesh according to the parameters the user defines in the Control Window. A .key file with the foundation layout is needed, to help the mesher get the shape of the foundation and mesh accordingly. As a result, a .key file is created.

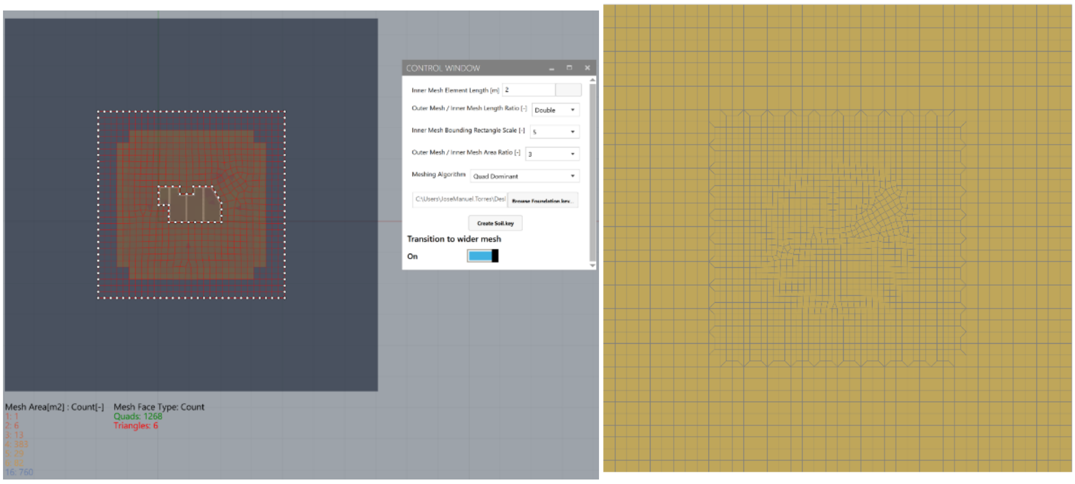


Figure 1 Rhino mesh (left) and the corresponding .key file (right) with the mesh transition.

4.1.2.2 Vertical transition

In the vertical direction, a practical way is to create a contact due to the non-coincident soil elements. The type of contact we normally use is *TIED_NODES_TO_SURFACE. An example of this contact card is depicted in section *4.4.5 Contact for vertical soil transition*.

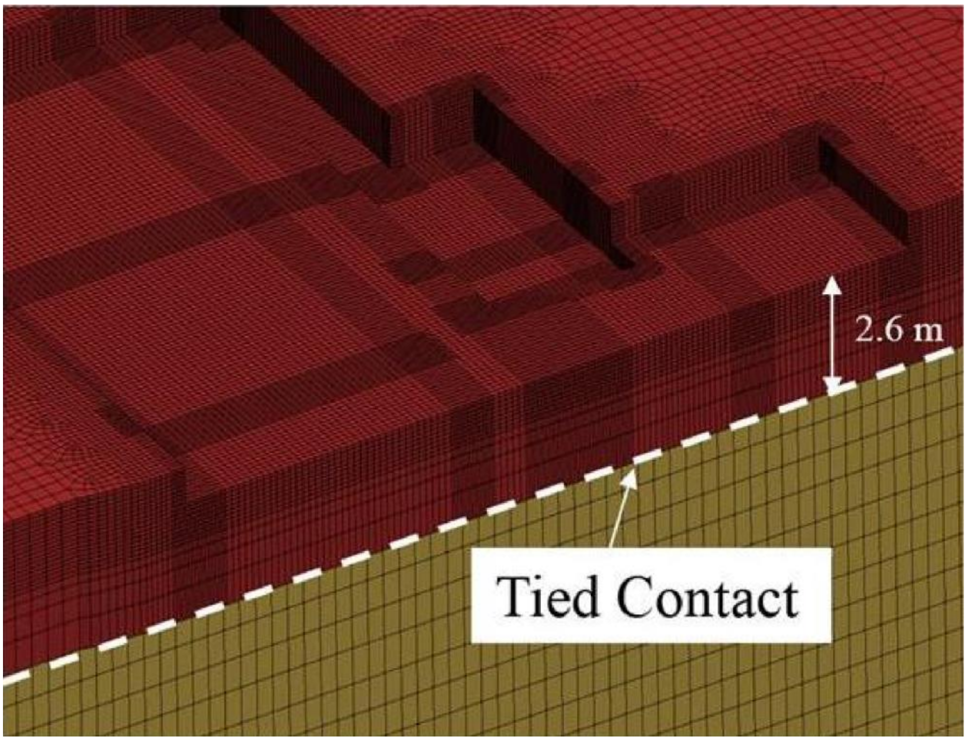


Figure 1 Soil contact at the corresponding depth

The script *layer_create_version_7_v02.js*, which extrudes shells in a .key file to create a soil block by reading a .csv file with soil properties, has been recently modified to allow for a contact depth definition.

$column	no							
$nrb	yes							
$lysmer_ro	2140							
$lysmer_vs	300							
$lysmer_vp	1490							
$lys_lcvx	11							
$lys_lcvy	12							
$lys_lcvz	13							
$tied_contact_z	-10.1							
zcoord	zbot	pid	mtype	stype	ro	k	lcur	lcd
0	-0.6	5000001	79	Clay	1682	4.06E+09	5000001	5001001
-0.6	-1.2	5000002	79	Clay	1683	4.06E+09	5000002	5001002
-1.2	-1.7	5000003	79	Clay	1683	4.06E+09	5000003	5001003
-1.7	-2.4	5000004	79	Clay	1479	4.06E+09	5000004	5001004
-2.4	-3.1	5000005	79	Clay	1479	4.06E+09	5000005	5001005
-3.1	-3.8	5000006	79	Clay	1479	4.06E+09	5000006	5001006
-3.8	-4.5	5000007	79	Clay	1479	3.79E+09	5000007	5001007
-4.5	-5.1	5000008	79	Clay	1479	3.79E+09	5000008	5001008
-5.1	-5.8	5000009	79	Clay	1479	3.78E+09	5000009	5001009
-5.8	-6.1	5000010	79	Peat	1173	3.77E+09	5000010	5001010
-6.1	-6.4	5000011	79	Peat	1173	4.43E+09	5000011	5001011
-6.4	-6.7	5000012	79	Peat	1173	4.43E+09	5000012	5001012
-6.7	-7.4	5000013	79	Sand	1887	3.77E+09	5000013	5001013
-7.4	-8.2	5000014	79	Sand	1887	3.77E+09	5000014	5001014
-8.2	-9.2	5000015	79	Clay	1683	4.50E+09	5000015	5001015
-9.2	-10.1	5000016	79	Sand	1887	4.27E+09	5000016	5001016
-10.1	-10.9	5000017	79	Sand	1887	4.27E+09	5000017	5001017
-10.9	-12.5	5000018	79	Sand	1968	4.27E+09	5000018	5001018
-12.5	-14.1	5000019	79	Sand	1968	4.27E+09	5000019	5001019
-14.1	-15.7	5000020	79	Sand	1968	4.27E+09	5000020	5001020
-15.7	-17.2	5000021	79	Sand	1968	3.97E+09	5000021	5001021
-17.2	-18.8	5000022	79	Sand	1968	3.97E+09	5000022	5001022
-18.8	-20.4	5000023	79	Sand	1968	3.97E+09	5000023	5001023
-20.4	-22	5000024	79	Sand	1968	3.97E+09	5000024	5001024
-22	-23.6	5000025	79	Sand	1968	3.97E+09	5000025	5001025

Tied contact depth

Figure 1 Snapshot from the input .csv file

Two shell surfaces must be defined (fine mesh and coarse mesh) and then the script will guide you through the process of extruding the fine mesh until reaching the depth of the coarse mesh.

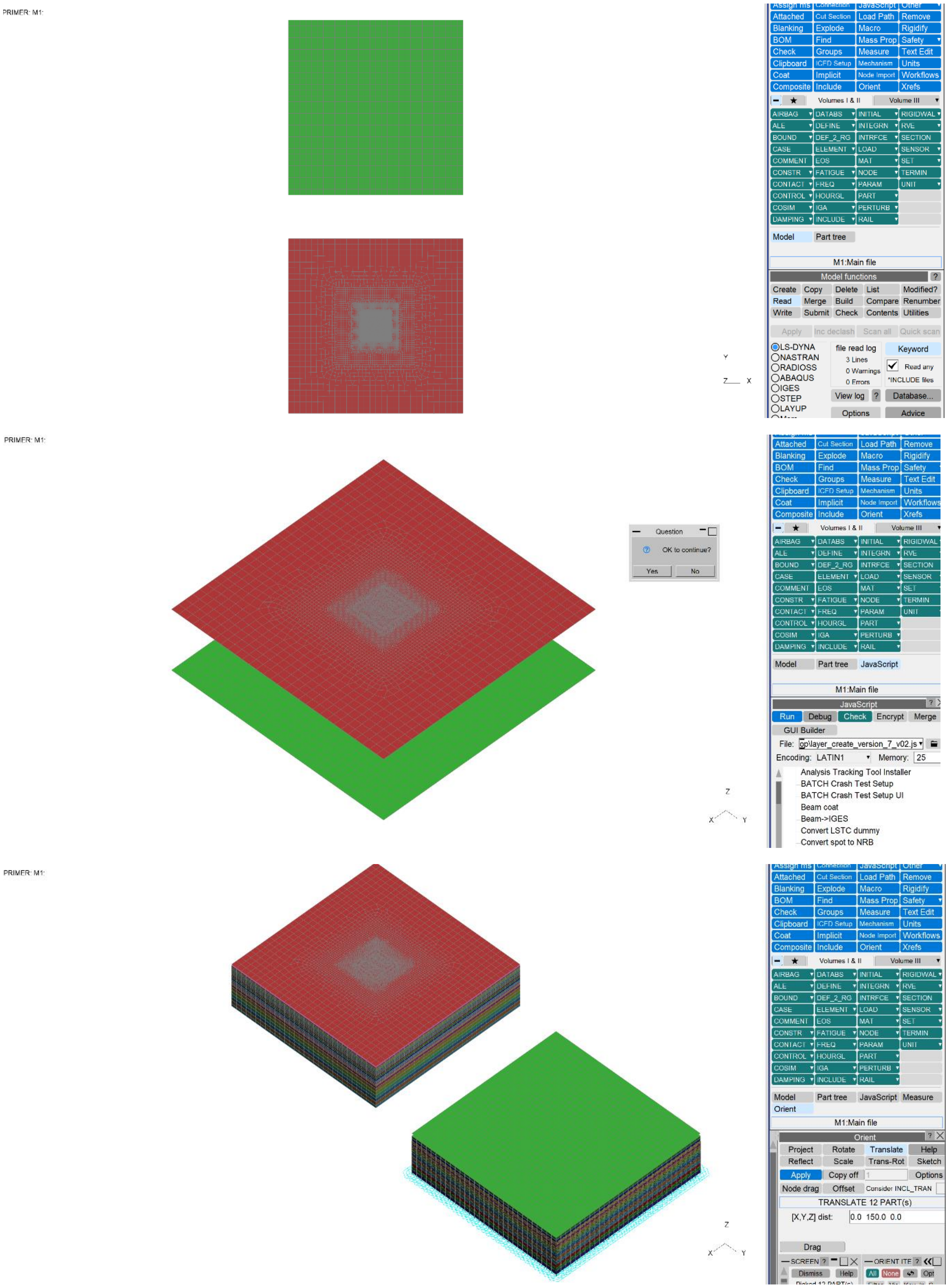


Figure 1 Steps in Primer to create the soil block with the vertical contact.

Reach out to Jose Manuel Torres for further indications on how to use and find these scripts.

4.1.2.3 Mesh calibration

When the mesh is fine enough to capture the bearing failure of the foundation, analyses should be carried out to determine the optimum mesh size to simulate such failure. An example of such a study (plane-strain test model) is shown in the images below. The soil mesh varies from 12 to 3 elements below the width of the footing. A compromise between accuracy and computational cost should be sought.

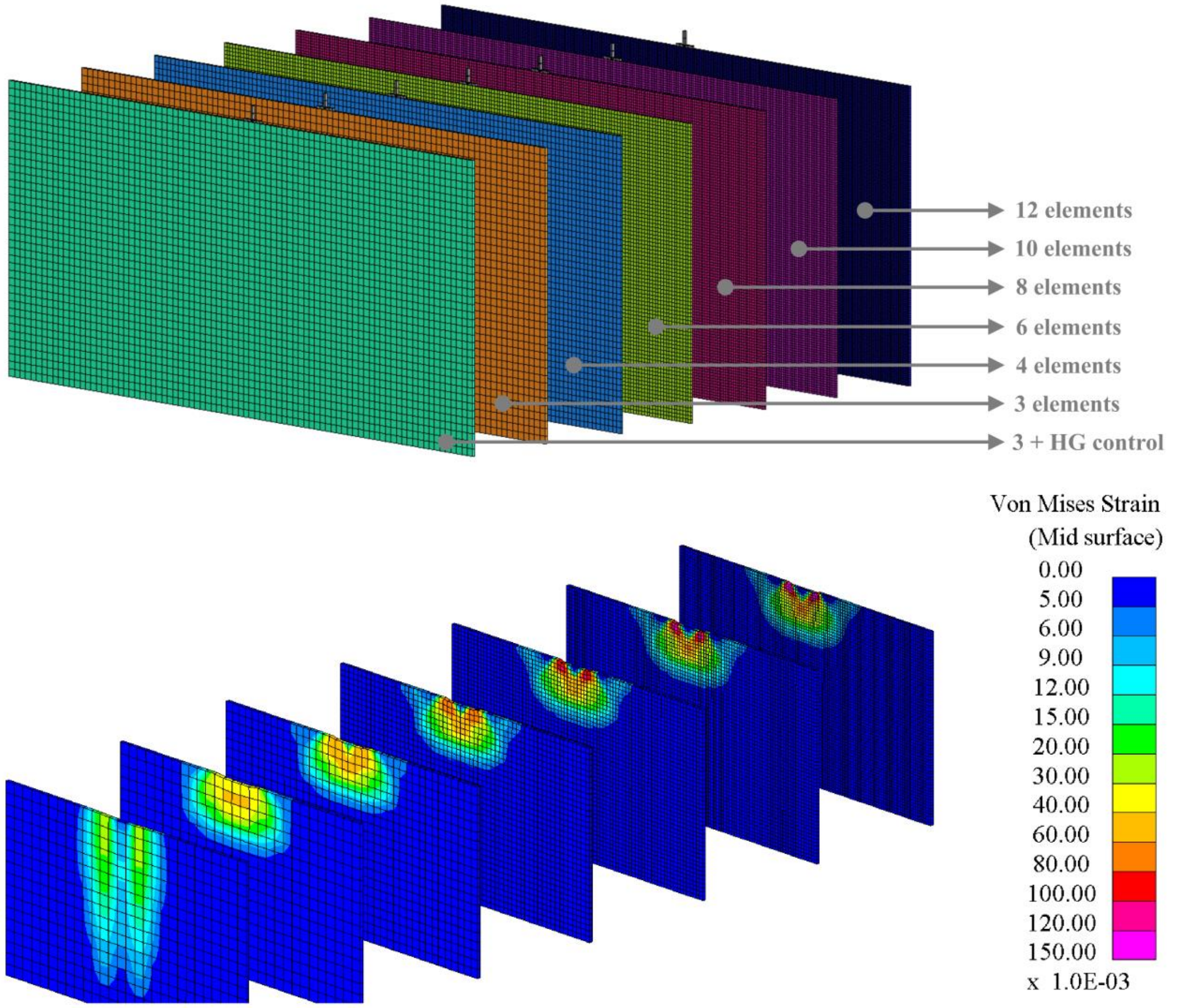


Figure 1 Mesh refinement to capture local bearing failure under a pad footing.

To carry out this test: 1) input a XYZ restraint along the base and 2) apply gravity first and then a pressure load or prescribed displacement to the top of the foundation slowly increasing until the footing has a bearing failure.

There are two ways to estimate the footing capacity from the LS-Dyna plane strain model. One is to apply a pressure loading to the top of the footing and look at shear strains in the soil elements, looking for the wedge failure mode in the output results graphically in D3Plot. You can then see what load was being applied at that point in the analysis. The more precise way to estimate capacity is to apply prescribed displacements to the top of the foundation. You can then plot principle compressive stress versus time directly under the footing to get the capacity as the peak principle compressive stress from that graph.

Also, in the modified direct approach (when modelling the soil mesh coarse enough to just capture nonlinear site response analysis and kinematic interaction and foundation flexibility is captured by soil springs), it is important to verify that the soil mesh is not adding any extra flexibility to the model, i.e. the mesh is coarse enough to mimic the effect of the spring connected to a fixed point. However, if it does add flexibility the mesh size, a calibration process needs to be carried out. Typically, it will consist in factoring down the spring stiffness to compensate the flexibility from the soil mesh (which is mesh dependent). An example of this calibration process for soil-pile springs is described in section *4.3.1.2 Pile-soil interface*.

4.1.3 Boundary conditions

The dynamic analysis of the unbounded soil domain is not trivial. When using finite elements to model the soil domain, we must make sure that the boundaries used do not introduce spurious reflections in the model that may alter the results of simulations. We have boundaries at the base of the soil block and at the lateral edges.

4.1.3.1 Soil block base

Absorbing boundaries are included at the base of the soil block. Each node is supported by a Lysmer damper represented using a discrete damper element (separate elements in x, y and z directions) to create a non-reflective boundary. This boundary allows wave propagations that are reflected from the surface to pass through the soil base and prevents the waves from bouncing back and forth between the soil surface and the soil base.

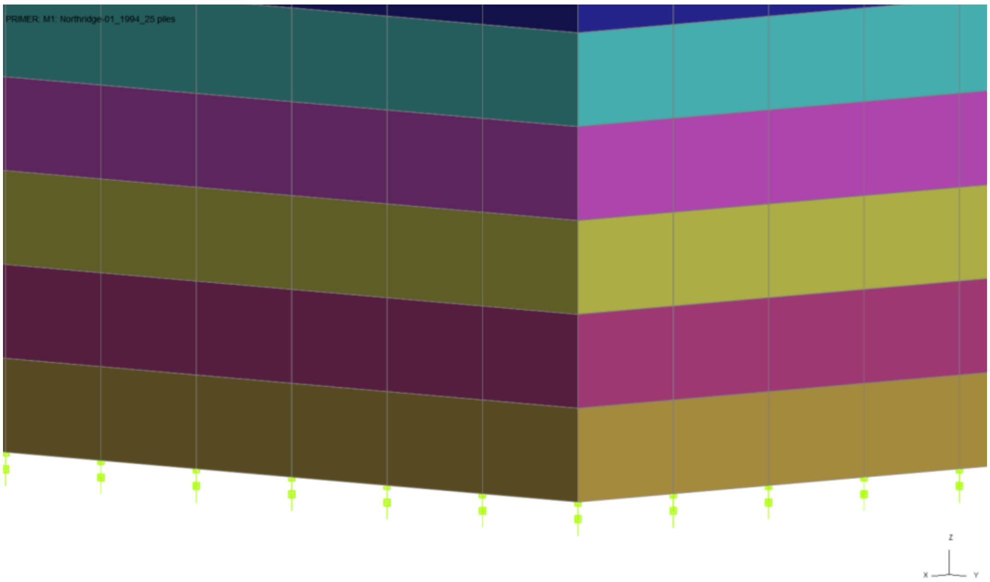


Figure 1 Lysmer dampers at the base of the soil block create a non-reflecting boundary.

Due to the nature of this type of boundary (dampers), the input ground motions must be applied in terms of ground motion **velocity time histories**. They are then multiplied by a scaling factor Δ in each direction, which converts the velocity time histories in force time histories.

$$F_x = \Delta_x \times V_x\,(t)$$

$$F_y = \Delta_y \times V_y\,(t)$$

$$F_z = \Delta_z \times V_z\,(t)$$

These factors depend on the tributary area of the element A, the material density ρ, the shear wave velocity V_s and the longitudinal wave velocity V_p of the bedrock stratum.

$$\Delta_x = A \times \rho \times V_s$$

$$\Delta_y = A \times \rho \times V_s$$

$$\Delta_z = A \times \rho \times V_p$$

Scaling factors in the two horizontal directions are thus equal. Typically for saturated soils V_p is taken as 1490 m/s.

An important fact to account is that the Lysmer dampers do not provide a support condition because viscous dampers provide a force that is proportional to velocity, not displacement. Therefore, the model will try to reach a quasi-static equilibrium (i.e. velocity approaching to zero) but will simply sink vertically and keep sinking. To overcome this problem, vertical loads counteracting the soil, foundation and superstructure gravity need to be applied during the analysis to keep the model in “vertical balance”. The process to get these forces in is detailed in section *4.4.1.2 Stop soil sinking*.

4.1.3.2 Soil lateral edges

A common strategy at Arup is to not include transmitting boundaries at the lateral edges and simply tie together the lateral boundary nodes without relative movement (i.e. with a nodal rigid body NRB between them).

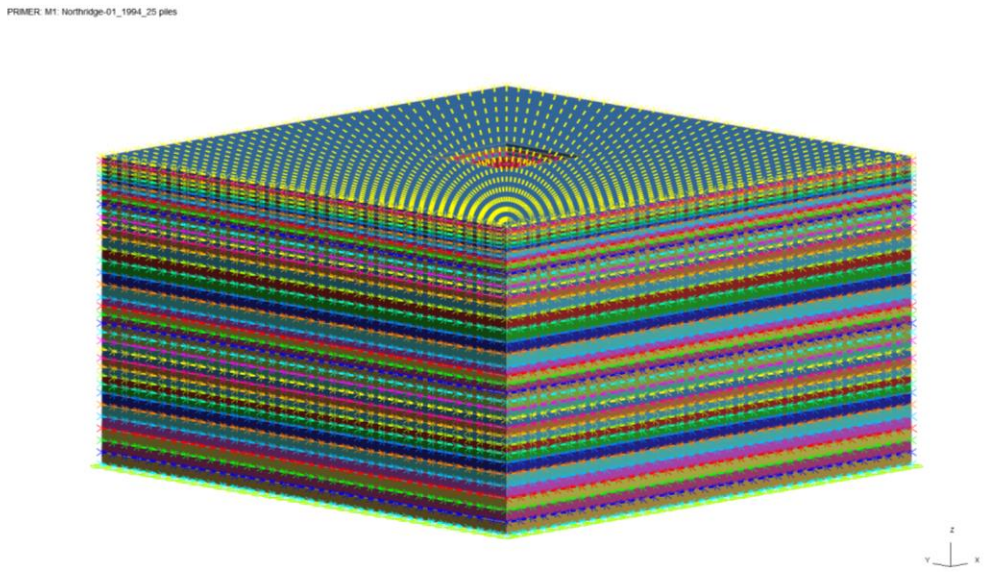


Figure 1 NRBs tying edge nodes at each soil level.

This is possible if we ensure a plan soil block size where the response of the soil at the corner of the soil block gives the same response as the single element column of soil subject to the same ground motion. In other words, this means the edge of the soil block is unaffected by the presence of the superstructure.

This can be verified by computing time history quantities (mainly displacements) or the free field surface response spectrum of the corner node and comparing them against the surface response spectrum from the Site Response Analysis.

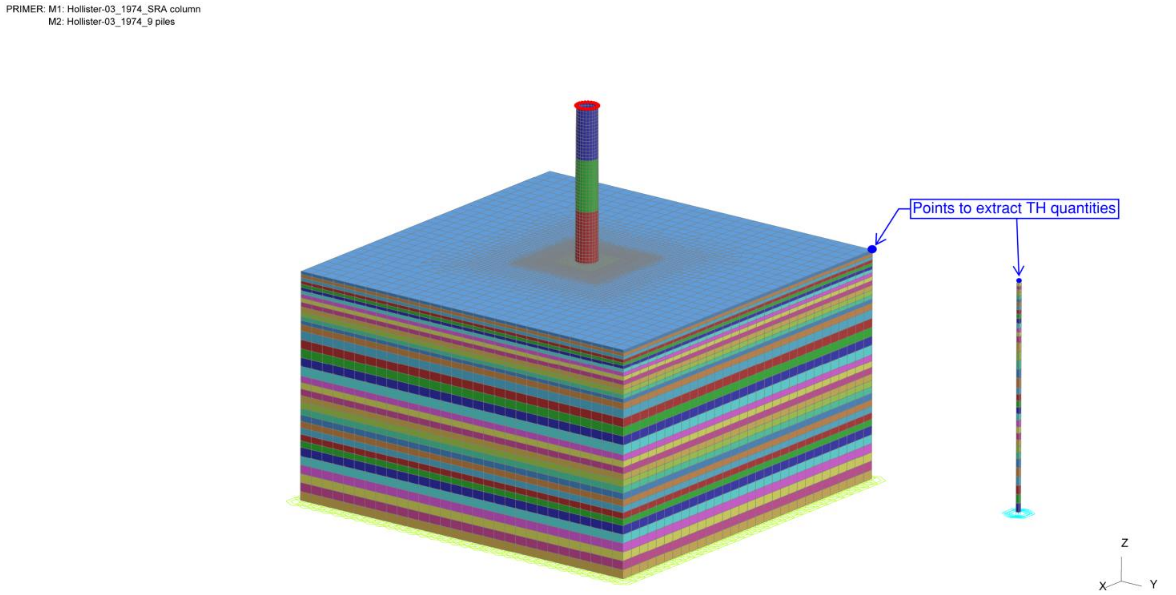


Figure 1 Corner point to check free field quantities against site response analysis output.

As a rule of thumb, previous experiences show that 4-5 times the footprint are normally enough to show the structure is not modifying the soil response. If this leads to a prohibitive soil block size, a careful analysis should be done to justify smaller sizes. SRA in LS-DYNA and Siren are automated, reach out to Jose Manuel Torres for further guidance on this.

4.2 Modeling of foundation

Different modelling options exist for foundation elements. Typically, shells, thick shells and solids are used. Also beam elements for piles are often used.

In Moodle there is an introductory course on how foundations are modelled in the P500 project. https://moodle.arup.com/course/view.php?id=4126

More information about coating thick shells to represent reinforcement and about typical material card definitions (e.g. *MAT_HYSTERETIC_BEAM for beam piles) will be included in subsequent versions of this guide.

4.3 Modeling of interfaces

Depending on the approach followed for the analysis of the SSI, the interfaces between the structural elements and the soil are modeled differently. This section describes the different modelling strategies for the interfaces according to the modeling method of SSI.

4.3.1 Direct Approach

Direct methods analyze the complete soil–foundation–structure system in its integrity. To this end, in Arup the soil and the structure are both modeled using finite elements. Within this method, discontinuities between structural and soil elements can be typically modelled with:

- Zero-thickness elements. (aka contacts)
- Thin layer elements. (aka elements with degraded properties, interface solid element).

Depending on the problem and the analyst's expertise, one option or another may be preferred.

4.3.1.1 Shallow element – soil interface

It is common to have a different mesh size in the foundation elements and in the soil around the foundation. To take the soil-foundation interaction into account, the *AUTOMATIC_SURFACE_TO_SURFACE contact is a good option. It can be defined for both horizontal and vertical surfaces.

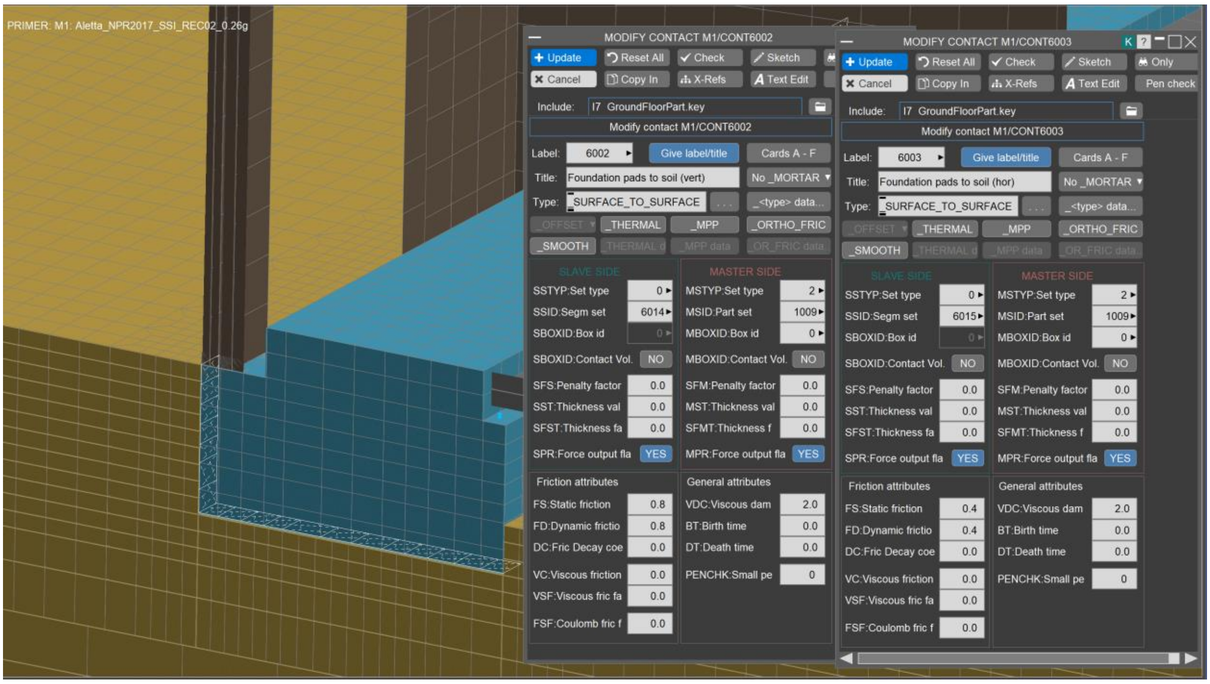


Figure 1 Contact cards for horizontal and vertical directions

Sometimes is also interesting to define an interface solid element with degraded properties to represent the disturbed soil around the interface between the structural element and the soil. This is very useful when congruent meshes are needed. It is usually a more stable solution than contacts.

Below an image of a circular monopile (red shell element) surrounded by interface element (blue solid elements on both sides of the monopile) and then the soil (green solid elements).

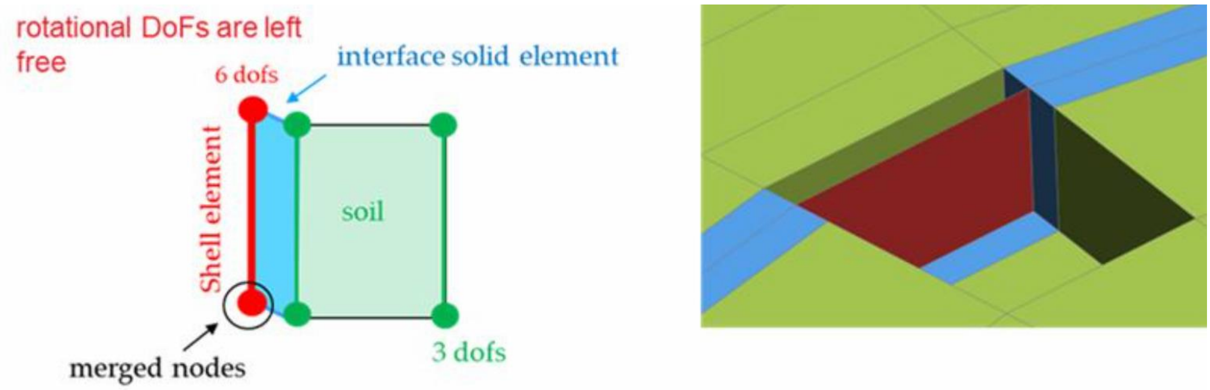


Figure 1 Thin layer element used to model the interface of a monopile with the surrounding soil.

4.3.1.2 Pile – soil interface

It is not very common to model the piles with solid elements, but sometimes may be needed. Again, *AUTOMATIC_SURFACE_TO_SURFACE contact is typically used both for tip and shaft interaction. This way we create a tensionless frictional interface along the shaft, which will allow gapping and relative sliding between the soil and the pile (if happening). The segment option is preferred for the soil (Master side). A segment set can be created: SET > SEGMENT > CREATE > COAT ELEMENTS and select the elements you want to coat.

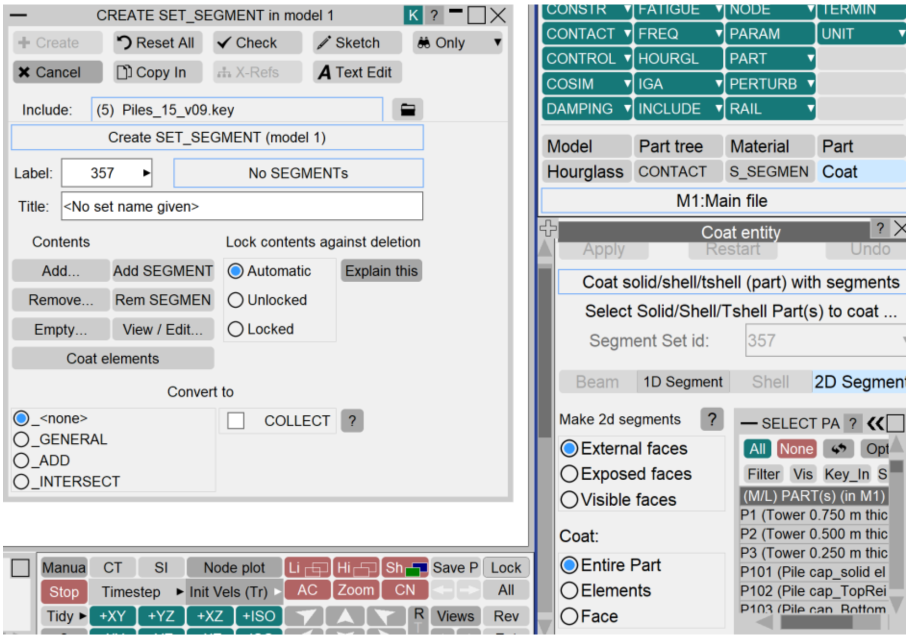


Figure 1 Coat solid elements to create segment within a segment set

For the pile shaft, a global frictional coefficient can be defined to match a desired shaft resistance. If known, it is possible to define a friction attribute per soil layer.

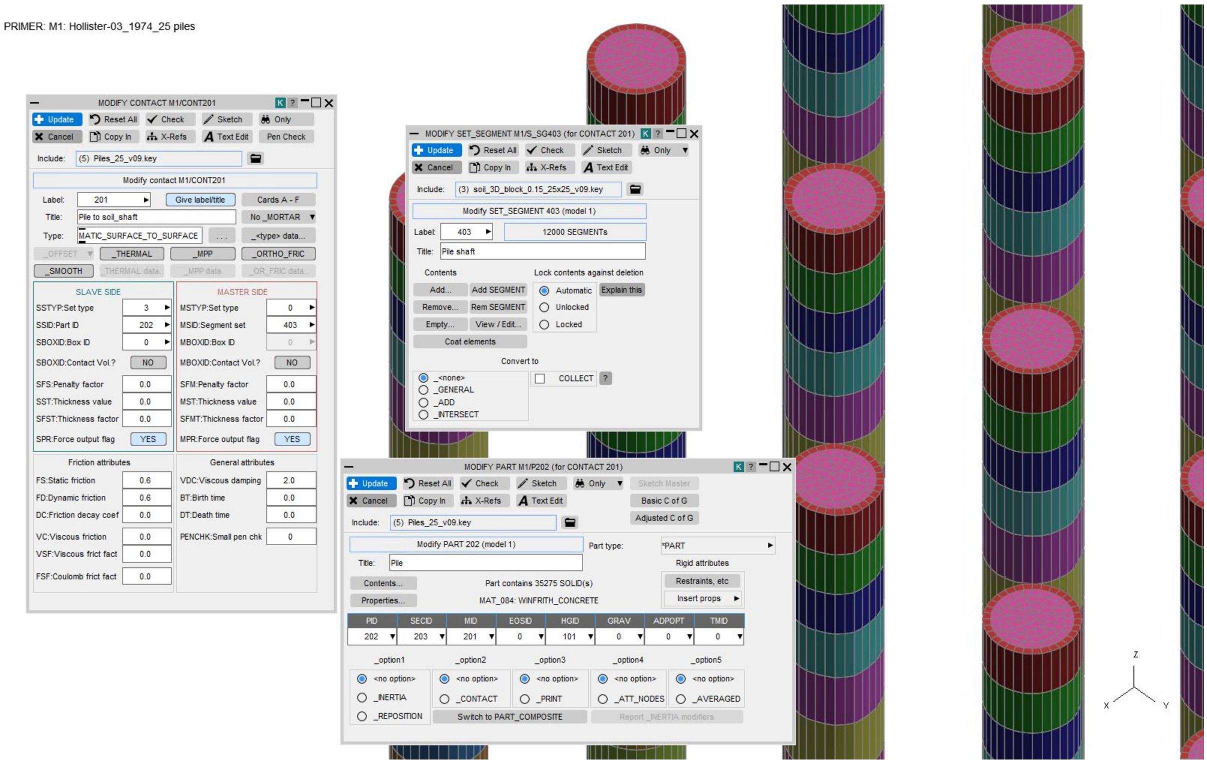


Figure 1 Example of contact card for shaft resistance

For the pile tip, the friction attribute is less important and the mesh refinement of the soil layer below it gains importance as it will influence the tip resistance. Therefore, a calibration might be necessary (similar to the one presented in section *4.1.2.3 Mesh calibration*).

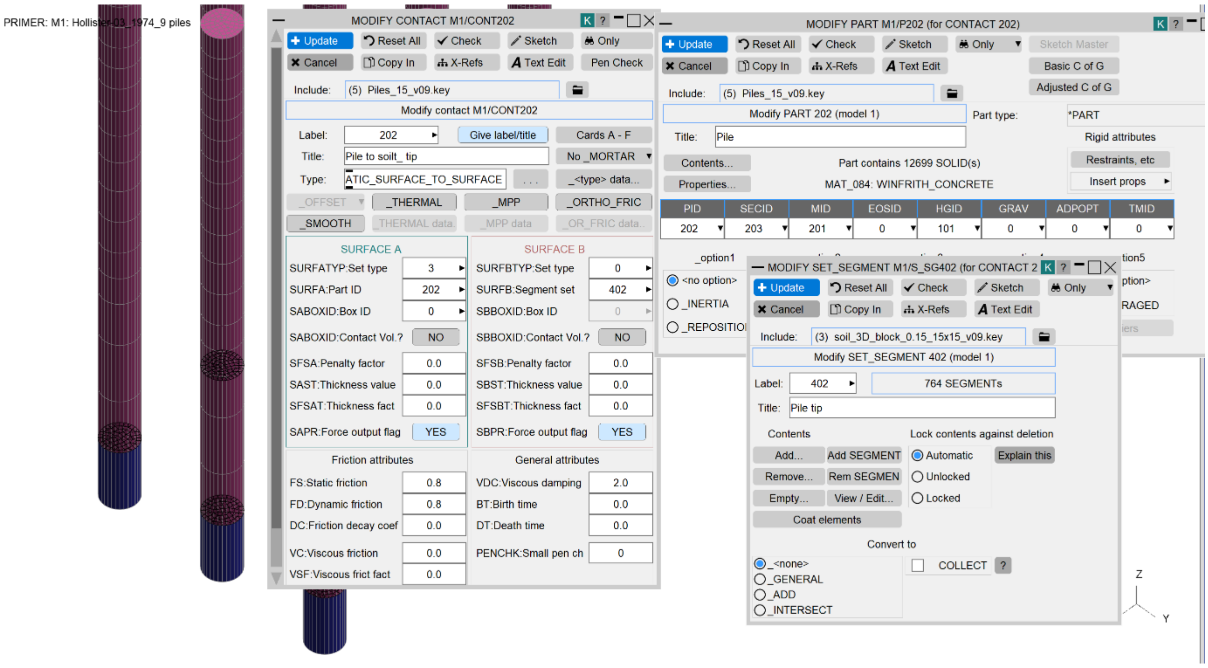


Figure 1 Example of contact card for tip resistance.

4.3.2 Substructure Approach

The substructure method decomposes the complete soil–foundation–structure system into several subdomains.

**Please note that this approach has not been very used in LS-DYNA and therefore tested under many conditions at Arup. Directly applying a ground motion at the lower node of a beam representing a soil spring is still under checking.*

4.3.2.1 Shallow element – soil interface

Typically, soil flexibility is represented with springs and dampers are included to capture foundation damping. Spring properties depend on the stiffness of the soil (shear modulus G), Poisson ratio, geometry and embedment of the foundation element and the excitation frequency. A typical formulation for shallow foundations used in past projects for defining the stiffness and damping values is the one proposed by *Pais and Kausel 1988*, which could be found from Table 2-2a to Table 2-2c in the NIST GCR 12-917-21 publication. This formulation is included in DesignCheck2:
https://compute.arup.digital/persistentCalcs/cd5fa549-8a9d-4e69-8920-690f3d4e85f7

In LS-DYNA, elements like footings, rafts, slabs, pile caps… can be supported by other elements. These supporting elements can be generally represented by discrete elements or beams.

Discrete elements: springs and dashpot can be defined as discrete elements (ELEMENT > DISCRETE) using the following material cards.

MATERIAL types
MAT_S01: SPRING_ELASTIC
MAT_S02: DAMPER_VISCOUS
MAT_S03: SPRING_ELASTOPLASTIC
MAT_S04: SPRING_NONLINEAR_ELASTIC
MAT_S05: DAMPER_NONLINEAR_VISCOUS
MAT_S06: SPRING_GENERAL_NONLINEAR
MAT_S07: SPRING_MAXWELL
MAT_S08: SPRING_INELASTIC
MAT_S15: SPRING_MUSCLE
MAT_S11: SPRING_BOLT_TENSION
MAT_S12: SPRING_BOLT_SHEAR
MAT_S13: SPRING_TRILINEAR_DEGRADING
MAT_S14: SPRING_SQUAT_SHEARWALL

Figure 1 Materials available for discrete elements

For springs *MAT_S01, *MAT_S03, *MAT_S04, *MAT_S06 are used depending on the degree of complexity to add in the model. For the damper, *MAT_S02.

<u>Beam elements</u>: another option is to define the supporting elements as beams with a tiny length (e.g. 0.01 m).

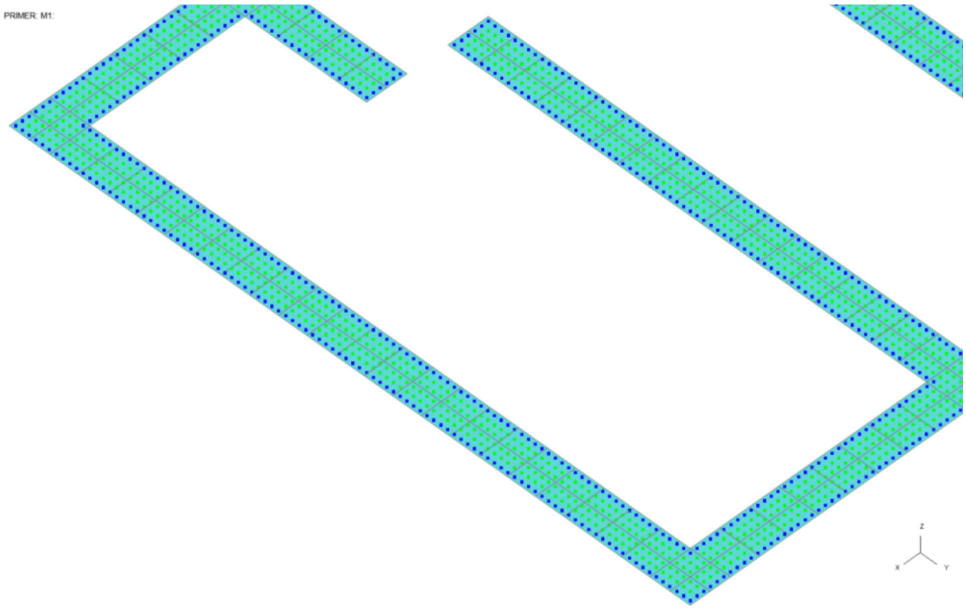


Figure 1 Example of beams distribution below strip footings

To define their properties, both stiffness and damping, material *MAT_205_P: DISCRETE_BEAM_POINT_CONTACT is used. It has the possibility to define in one single card:

- Nonlinear load-displacement curve. (LCZ)
- Vertical and horizontal stiffnesses and damping coefficients. (STIFF & DAMP)
- Upper limit on friction force. (FRMAX)
- A gap in Z direction, usually used to control the load share between pile caps and piles. (GAP0)

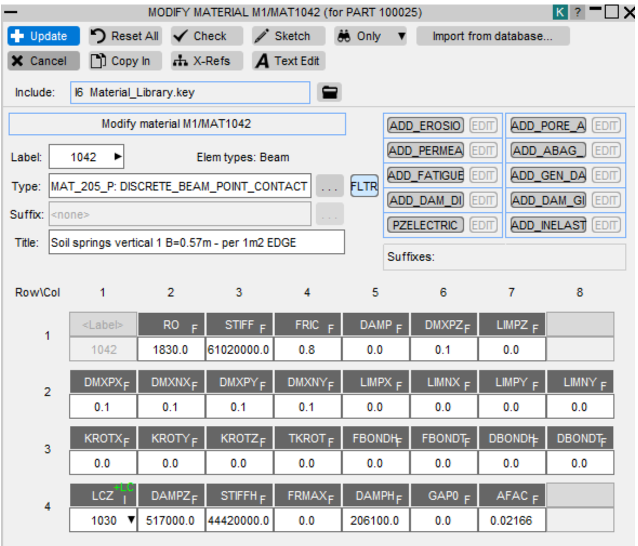


Figure 1 Material card used for Material 205_P

A common approach is to define the material properties per $1m^2$ and then use the AFAC value to input the tributary area of the element. If horizontal springs are to be included, just be aware that Z is the axial direction of the beam element, therefore always perpendicular to the soil face. It is recommended to model foundations with solids or thick shells if horizontal springs are incorporated to the analysis.

For further guidance on how to populate this material card, please reach out to Jose Manuel Torres.

A contact must be defined to link the upper nodes of the beams with the shell elements defining the foundation element. This is achieved with a *TIED_NODES_TO_SURFACE contact where the nodes are tracked to the part set containing the foundations.

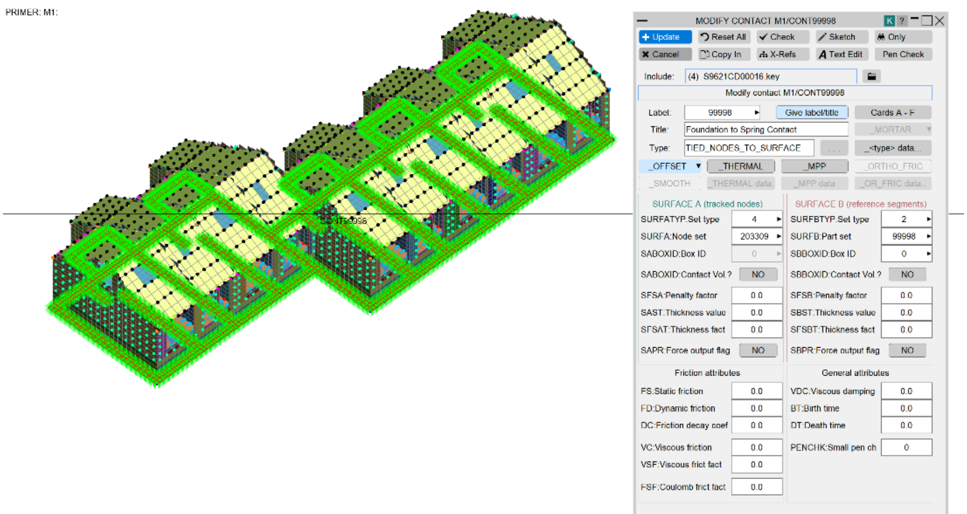


Figure 1 Contact definition between shell foundation elements and upper beam node

The ground motions are applied at the lower node of the beams as **acceleration time histories**, and it is necessary to check that the full motion is transmitted upwards to the structure.

4.3.2.2 Pile – soil interface

In the case of piled foundations, the springs and dashpots are not attached to the foundation element (piles) as it was the case in the previous section, where we defined a contact to attach springs to the shallow elements. With piles, the dynamic impedance functions replace the piles in the numerical modeling.

The simplest option for replacing a pile or a pile group would be to model a beam under the pile cap/grade beam and use the *MAT_205_P: DISCRETE_BEAM_POINT_CONTACT material card to represent the 3 translational and rotational stiffnesses of the elements replaced.

4.3.3 Modified Direct Approach

This approach has been used extensively in the P500 project both for shallow and deep foundations. It is a way of having all the elements in a single model but without the computational cost of the direct approach.

4.3.3.1 Shallow element – soil interface

The process is very similar to the one for the substructure approach. Springs and dashpots are included as small beams underneath the foundation element. The material card is again *MAT_205_P: DISCRETE_BEAM_POINT_CONTACT

The beams are connected to the foundation element (typically shells) and to the soil elements (solids). In both cases the idea is to tie the beams nodes to the shells or solids. Thus, a TIED_NODES_TO_SURFACE contact is a common solution.

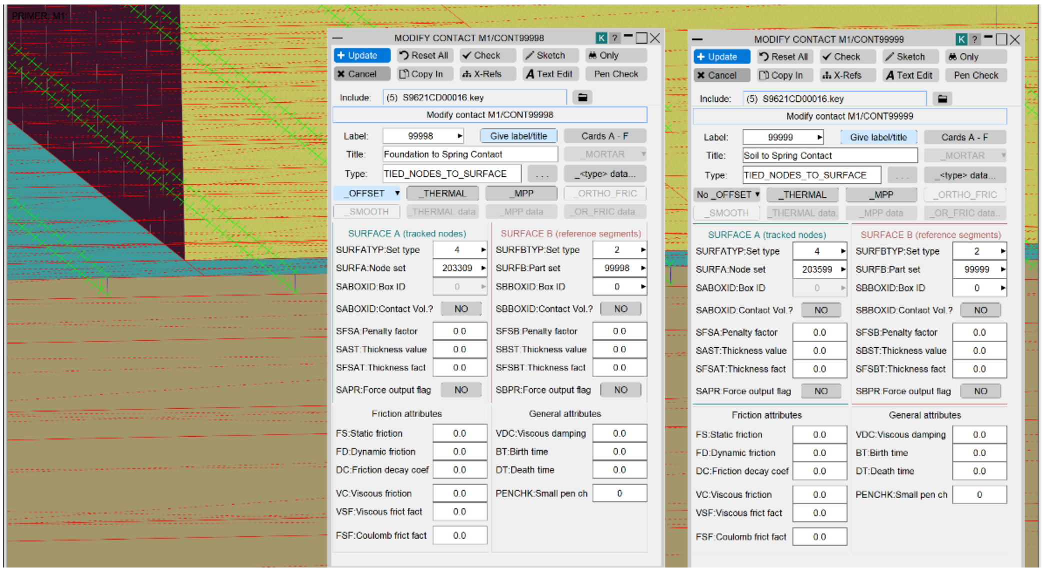


Figure 1 Example of TIED_NODES_TO_SURFACE contact for upper and lower nodes

With this approach, and to avoid a calibration, typically the soil mesh is chosen coarse enough to not add any additional flexibility. Below it is depicted a good example, with a coarse mesh with around 2x2m elements below the building. This way, the typical formulations for spring and dashpot coefficients can be used without further modification.

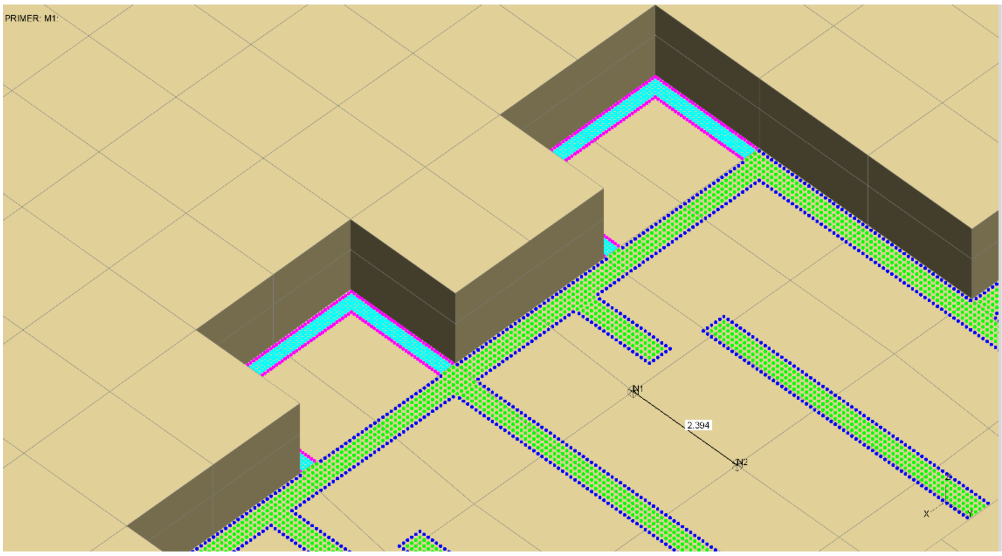


Figure 1 Coarse 2x2m mesh underneath soil springs

4.3.3.2 Pile – soil interface

Pile-soil interaction is commonly defined with springs linking piles modeled as beam elements with the solid elements of the soil. These are defined in LS-DYNA with the*CONSTRAINED_SOIL_PILE card.

Generally, these springs pick up the soil-pile flexibility. However, sometimes the soil block provides some flexibility (which is mesh-dependent), so a calibration process is required so that the spring and soil provide the desired total flexibility. As an example, the typical steps for calibrating CSP cards (soil-pile springs) connecting a pile element to a *slightly* coarse soil mesh are as follow:

1. Obtain lateral p-y springs from ALP.
2. Obtain vertical side friction and end bearing capacities (and sometimes t-z/Q-z springs) from spreadsheet calculations or literature.
3. Use peak capacities vs depth from #1 and #2 to assign BLCD, VLCD and HLCD.
4. Either use half of the normalized p-y/t-z/q-z spring curves or some other simplified assumption (like very stiff for horizontal springs, ¼ inch to full mobilization for side friction, ½ inch to full mobilization for end bearing) to define BLC, VLC and HLC
5. Perform a free-head or fixed head lateral pushover analysis and compare with equivalent ALP result. Adjust #4 if necessary to get a good match.
6. Perform a vertical push analysis and compare with t-z analysis. Adjust #4 if necessary to get a good match.

There is a more user friendly CSP option, called **CSP_CURVES**, developed internally at Arup. One of the advantages of this option is that allows to decoupling with depth the behavior in each direction and is easier to fill in.

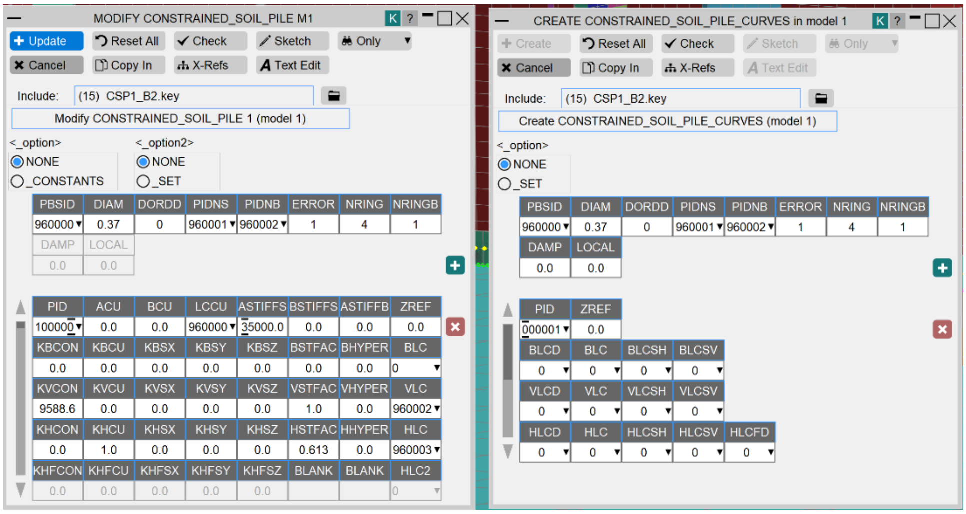


Figure 1 CSP classical option (left) and CURVES option (right)

As a minimum, it requires two curves for each direction: one defining the peak capacity of the soil (BLCZ, VLCZ and HLCZ) and the other being the "normalized mobilization curve" (BLC, VLC and HLC). These curves are multiplied in each direction to get the coupling stresses for base, axial and perpendicular coupling.

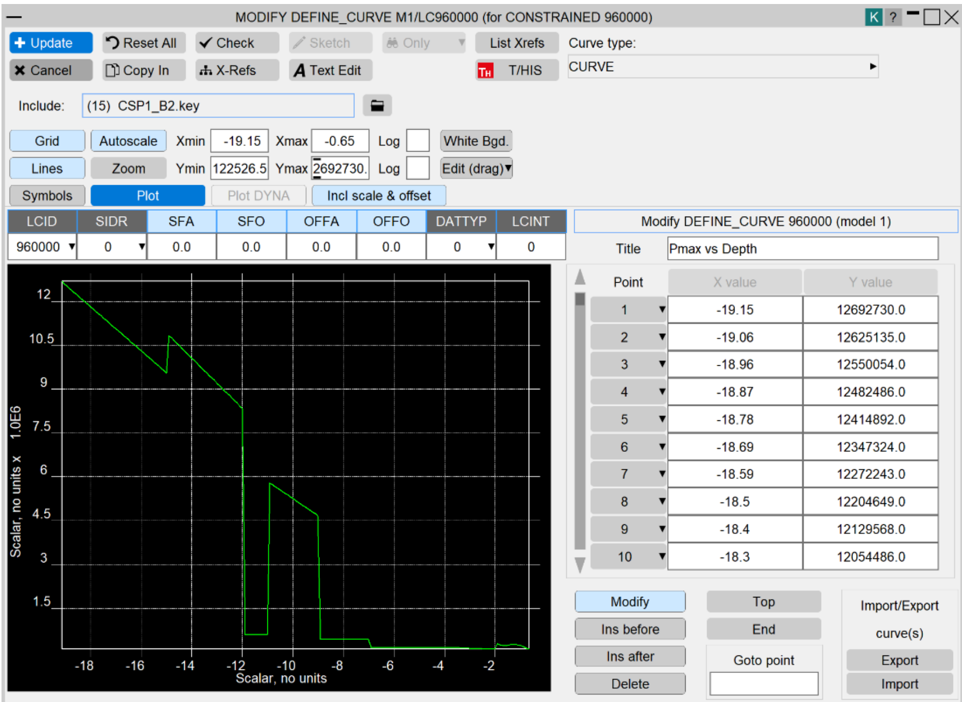


Figure 1 Example of a peak capacity curve vs depth (VLCZ)

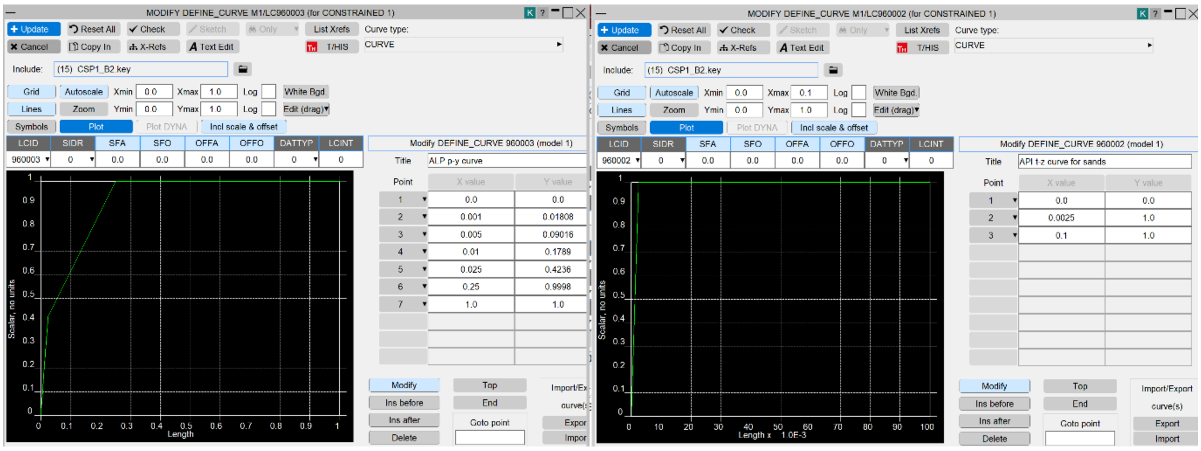


Figure 1 Example of HLC (left) and VLC (right) curves.

If the soil domain is variable or you want to capture a capacity that varies during the simulation, you may use VLCSH and BLCSH instead of VLCD and BLCD. But for regular applications, this is not necessary.

To conclude, it’s worth mentioning that the CSP_CURVES option is recommended, especially if new automation tools are envisaged. There is already an automation which populates the traditional and less intuitive CSP card. It is currently embedded in a wider geo-seismic workflow, but it is expected to be extracted soon and make it more independent to ease its use. Reach out to Jose Manuel Torres for more information about this tool.

4.4 Watch-its

4.4.1 Free field motion sanity check

As mentioned in section *4.1.3.2 Soil lateral edges*, the horizontal extent of the soil should be enough to dissipate the effect of the structure. To this end, the free-field surface response spectra at a soil block corner should be compared with the one from a site response analysis. The SRA can be performed either with Siren, DeepSoil or LS-DYNA too. LS-DYNA is the only option for running the 1D SRA with bi-directional shaking.

It is recommended, depending on the thickness of the soil elements, to pass a Butterworth filter to the acceleration signals (THIS > Automotive > BUT) to remove numerical noise if present.

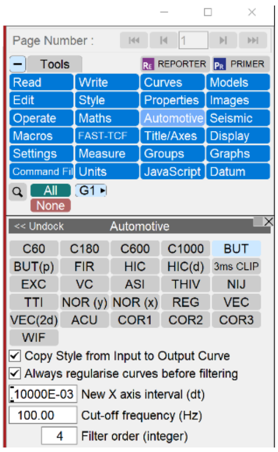


Figure 1 Butterworth filter command in THIS

4.4.2 Stop soil sinking

An important fact to account is that the Lysmer dampers do not provide a support condition to the soil block because viscous dampers provide a force that is proportional to velocity, not displacement. Therefore, the model will try to reach a quasi-static equilibrium (i.e. velocity approaching to zero) but will simply sink vertically and keep sinking. To overcome this problem, vertical loads counteracting the soil, foundation and superstructure gravity are applied to each base node. The following steps are taken:

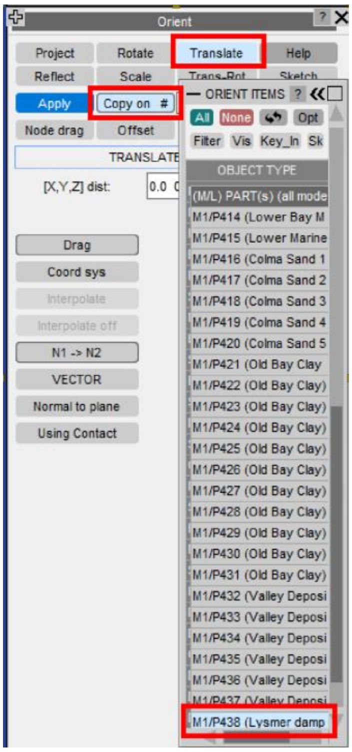
<u>Step 1: Stiff Springs Model</u>

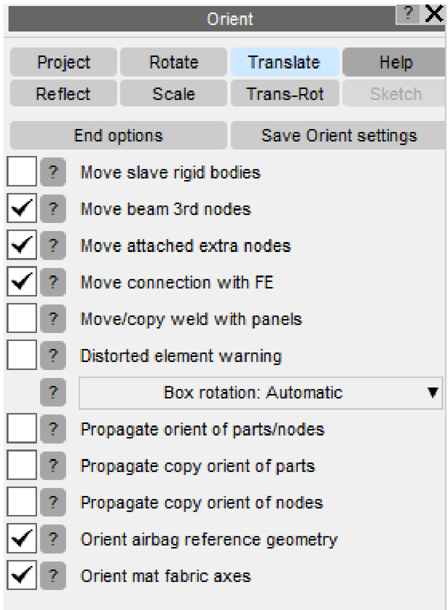
Set up the soil block and superstructure model with all relevant control cards and gravity applied to the parts requring it (sometimes gravity is not applied to the soil block just to the superstructure, as in P500 project).

Create stiff springs to stop the model sinking. The spring elements are identical to the Lysmer dampers (same nodes, orientation vectors) except they have different material properties. The required steps can be as follows:

i. Open the soil block include file in Primer.

ii. Go Tools Menu > Orient > Translate > set all translate distances to 0.0 >Object type – Part > Lysmer damper part > Copy on > Options > “elems in new part”, “create new sects/matls”, “parts, elems, nodes only”, “copied items to same include” > End options > Apply > Accept.





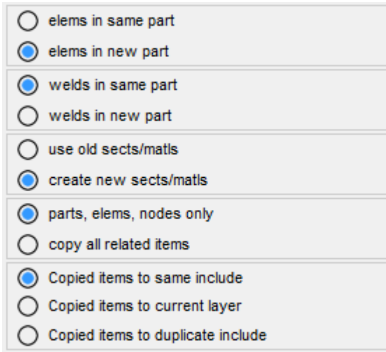


Figure 1 Options in Primer to be selected.

iii. Go to Part keyword keyword and select the new part created (it should be at the end). Change the title to “Stiff Springs”. Right click on MID > create. Select MAT_S01. Input a K of 1000.0 N/m.

iv. Merge the duplicated nodes.

v. The bottom of the soil block will then look like this, with both stiff springs and dampers at each node.

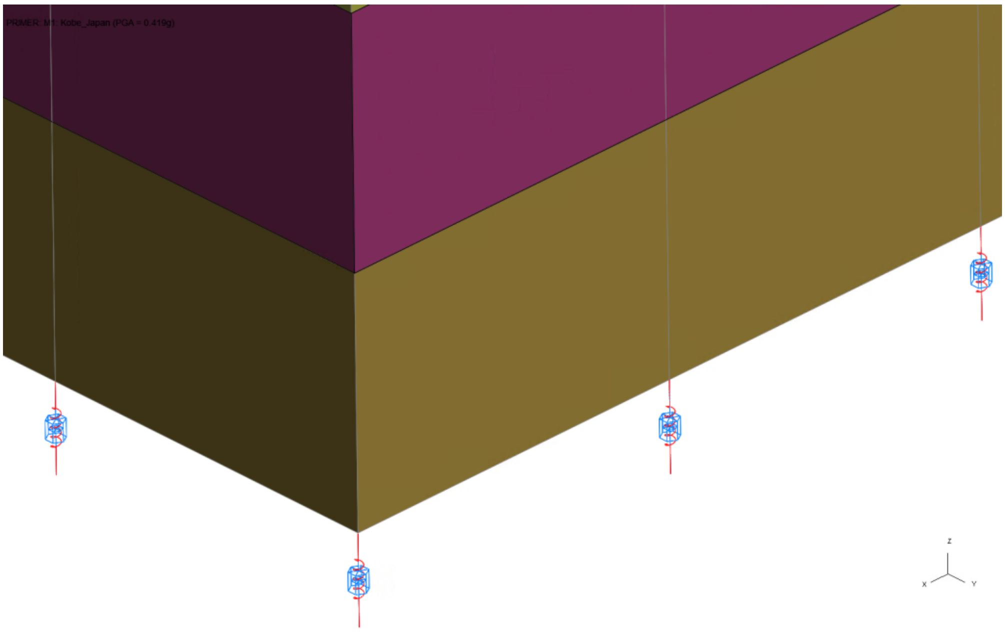


Figure 1 Bottom of soil block, showing Lysmer dampers and stiff springs in parallel.

vi. Run the gravity phase of the analysis

Step 2: Extract Spring Forces using D3Plot script

Extract the forces from the stiff spring elements. To do so, open the model from Step 1 in D3Plot (typically ptf file, but also the rlf file). Run the javascript: extract_spring_forces_D3PL_12_1.js

It will ask you which part contains the stiff springs. The output from the script is a file *LOAD_NODE.k*, containing the forces from the spring elements. Each force will be given twice: once for dynamic relaxation (using the curve that ramps up from zero) and once for the transient (using the curve that is always equal to 1.0).

Now you can also just define one curve for applying the forces: a first part matching the gravity ramping up and a second part constant for the transient analysis. In the example below, gravity for all the elements is applied during 1s ramping up in the first half, while the vertical loads increase linearly until 0.9s and then remain constant. This solution gave a stable model and a negligeable soil block constant Z displacement during the analysis.

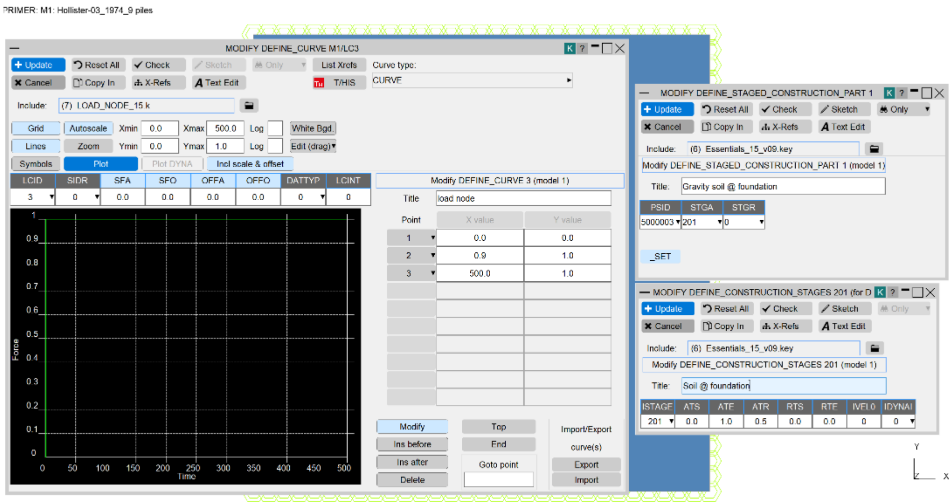


Figure 1 Bottom of soil block, showing Lysmer dampers and stiff springs in parallel.

<u>Step 3: SSI analysis</u>

Replace the stiff springs by the nodal loads reacting the gravity (i.e. delete the stiff springs part and add the *LOAD_NODE.k* include file to the master .key file).

4.4.3 Hourglass card

LS-DYNA has several default formulations to avoid or at least minimize hourglass modes. It is worth saying that before using any formulation, refining the mesh or switching to fully integrated element formulations can help in the hourglass battle.

A typical HG card for soil and solid RC elements is shown below.

Figure 1 Example of HG control card for soil and foundation elements.

Always check the hourglass energy in THIS. The very end goal is to keep hourglass deformation and hourglass (resisting) energy low.

4.4.4 Fix crossed edges in contacts

Sometimes, when a very refined mesh is defined at both sides of a contact, there could be penetrating and crossed elements. If this happens, detachment from the contact is not allowed and therefore is very important to always check if crossed edges are present in any contact, otherwise the contact won't work as expected and could lead to unrealistic constrains during the analysis.

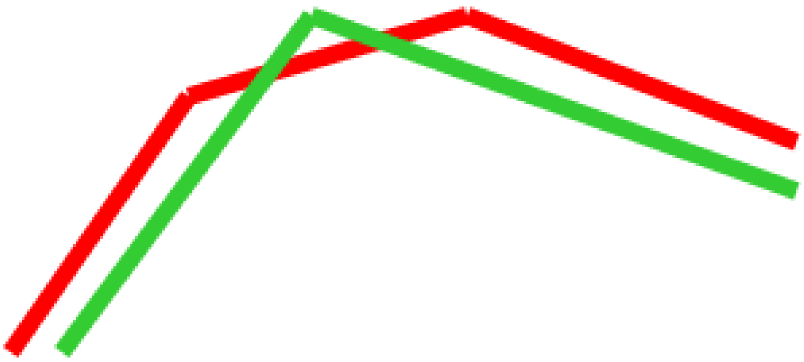


Figure 1 Schematic definition of a crossed edge

If the green line is a pile and the red line is the soil around it, this crossed edge will prevent the pile from separating from the soil and will create an unrealistic link between them.

In the contact card, if you go to Pen Check and tick List crossed, you will see the parts being crossed and the number of segments crossing. In the image below, parts 101 and 403 are crossing in 348 points.

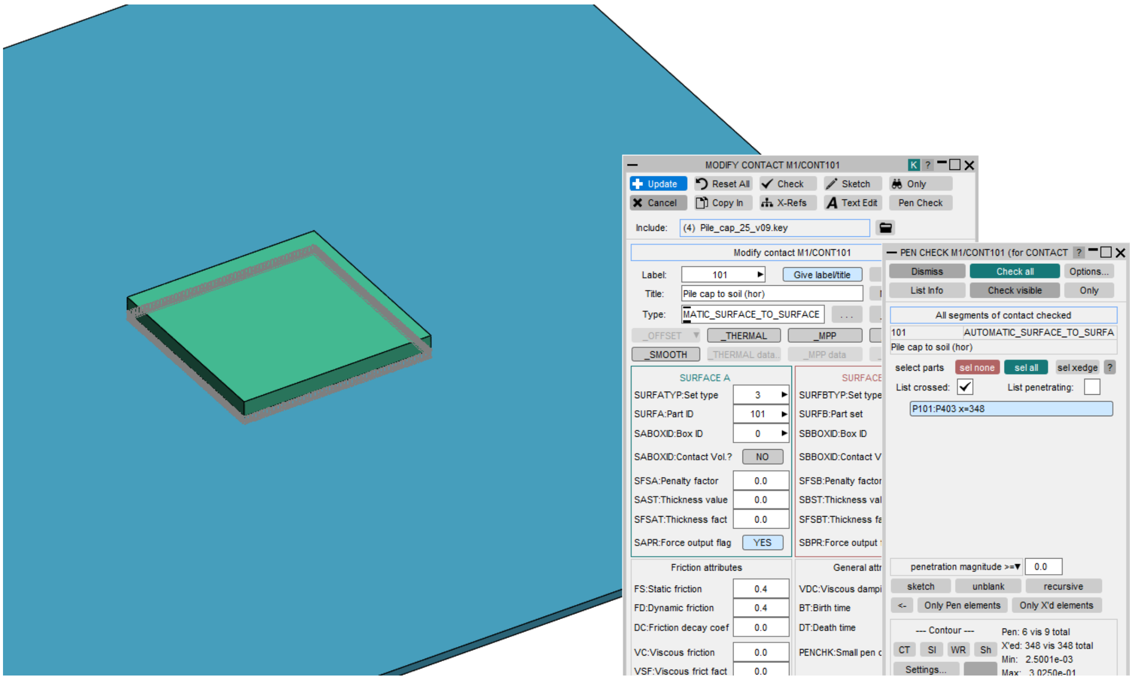


Figure 1 Nodes includes in the contact (left) and contact card for the soil tied contact (right)

A quick solution would be simply to adjust the geometry accordingly by moving the nodes crossing the other element.

4.4.5 Contact for vertical soil transition

Care must be taken to not include the lateral nodes in the contact as they belong to the NRBs which avoid relative movement between them.

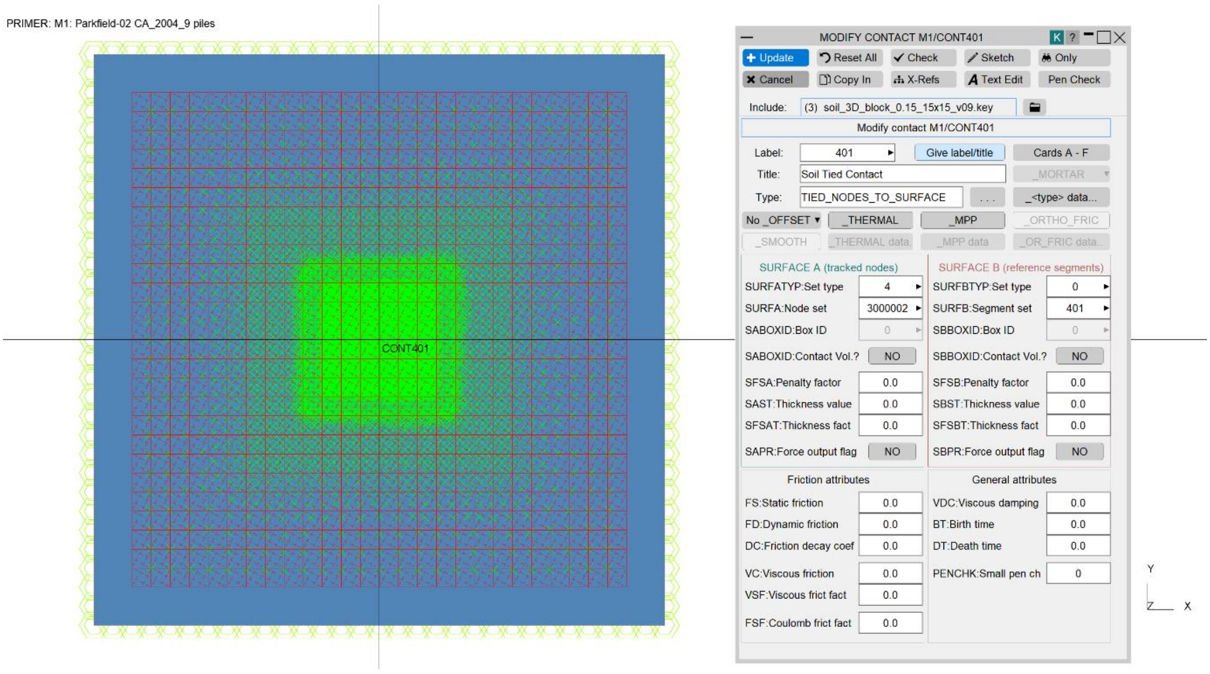


Figure 1 Nodes includes in the contact (left) and contact card for the soil tied contact (right)

4.4.6 Liquefaction modelling

When liquefaction is expected, *MAT_SOIL_SANISAND could be used to explicitly model the onset and the effect of liquefaction during seismic analysis. (same text as in the previous version of the guide could be used).