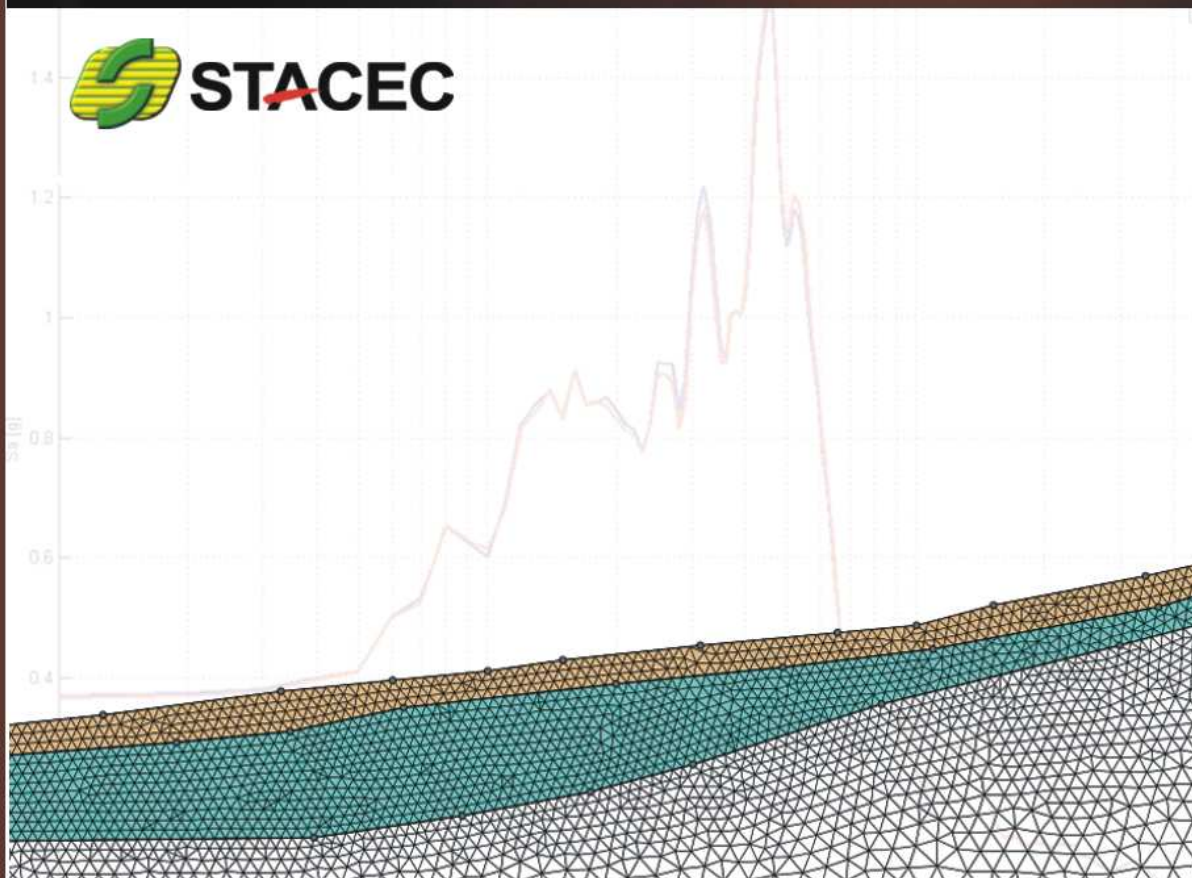
USER
MANUAL

LSR2D

STACEC



www.stacec.com

COPYRIGHT

For more information:

Stacec s.r.l.
Software & Services for Engineering
S.S. 106 – Km 87
89034 – Bovalino (RC)

Tel. 0964/67211
Fax. 0964/61708

USER MANUAL

Local Seismic Response 2D

1. Introduction

The characterization of soil motion at ground surface after a seismic event is an issue of extreme importance for structural design in seismic zones.
When no constructions are present on the ground, soil behaviour after an earthquake can be described by three mechanisms: source mechanism, wave propagation from source to site and local seismic response.
The last one is responsible of the modifications in terms of amplitude, time duration and frequencies, which affect seismic motion of a rock formation (reference earthquake), when it goes through layers from bedrock to the ground [Lanzo G. et al.,1999].

Local seismic response phenomenon has been dealt with in many researches for the last three decades. Those researches have been based on analyses of seismic events registrations and numerical modeling. Lately the phenomenon has been included into building codes.
Last italian seismic code ("Norme tecniche per le costruzioni, 2008, NTC2008 , § 3.2.2) proposes, a numerical approach (monodimensional or bidimensional) to evaluate real seismic response of soil layers using single building scale or single geotechnical system scale.

Numerical analysis for the local seismic response, is more accurate than the simplified approach proposed by NTC2008 (soil categories), and let you identify layer amplification, by finding period ranges more limitated than by using code approach and consequently you'll have a structural design which is economically better.
Using LSR 2D (Local Seismic Response 2d) software by Stacec you can model the problem in a 2D geometrical domain by a finite element approach, in time domain, total stress, with a Kelvin-Voigt soil model.

2.1 Graphical User Interface

Software main window is showed in figure 2.1. From this you can reach all the windows to define your model.

LSR2D graphical user interface is characterized by a **main window** and other secondary windows. Main window contains some menu from which you can access al functionalities.

Right part of the main window is the **Operative Area**.

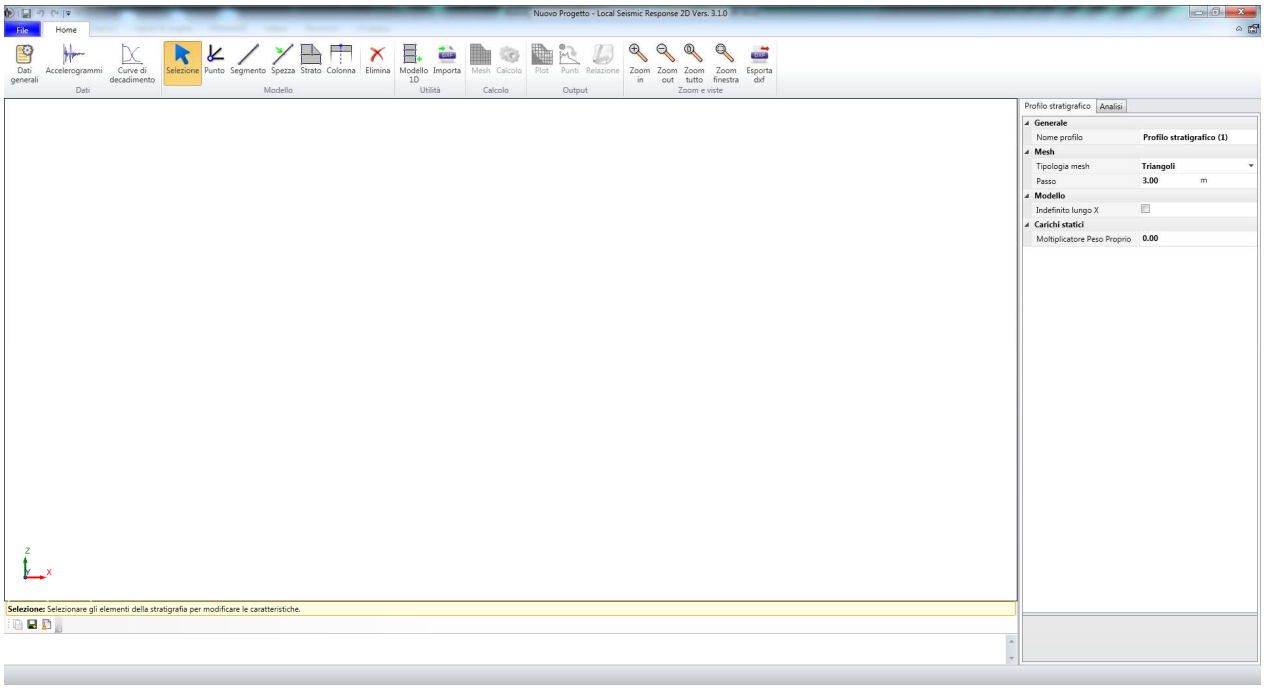
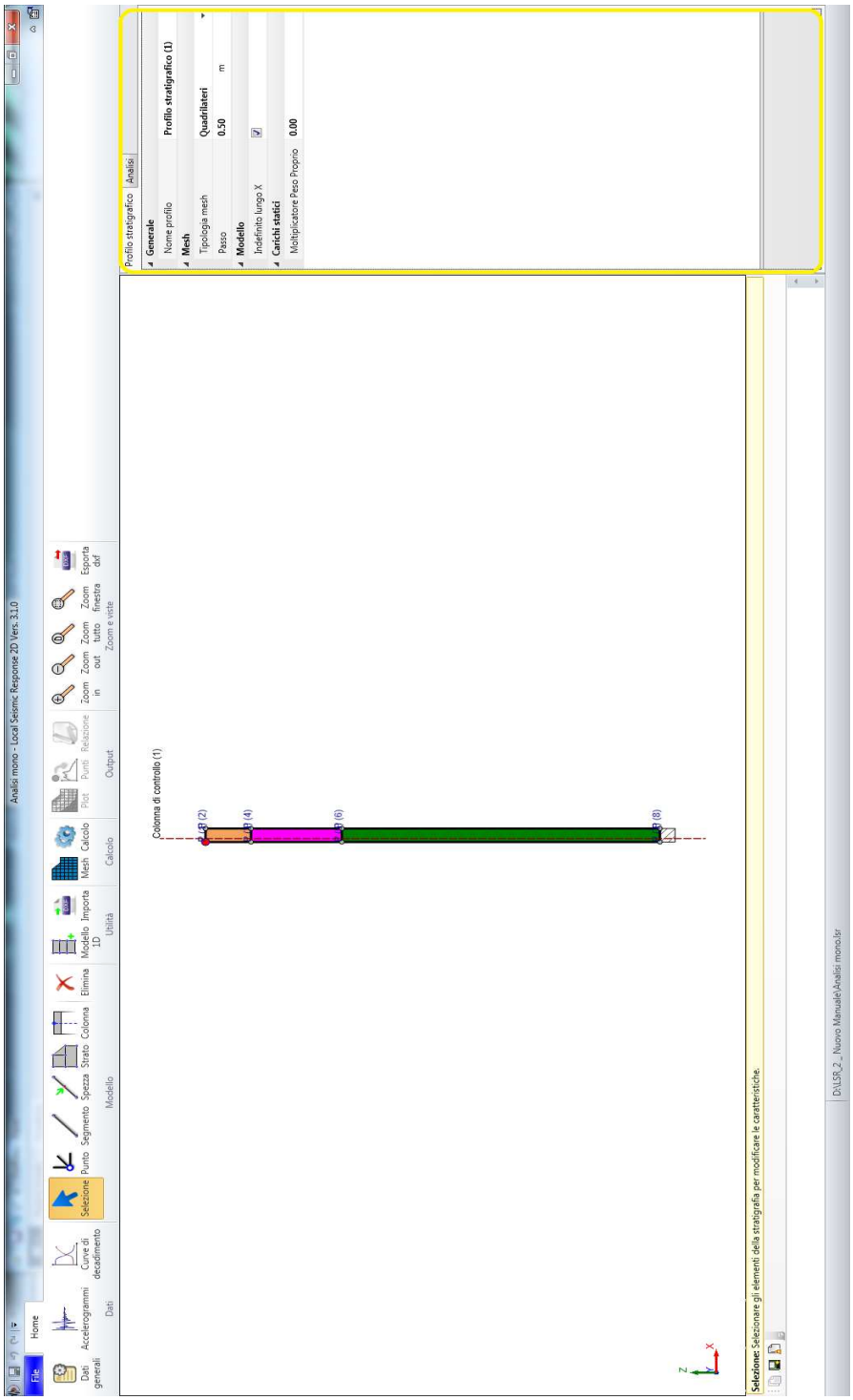


Fig 2.1 - Main window


Analisi mono - Local Seismic Response 2D Vers. 3.1.0
Home
File
Dati generali
Accelerogrammi
Curve di decadimento
Dati
Selezione
Punto
Segmento
Spezza
Strato
Colonna
Elimina
Modello
Modello 1D
Importa
Utilità
Mesh
Calcolo
Calcolo
Plot
Punti
Relazione
Output
Zoom in
Zoom out
Zoom tutto
Zoom finestra
Zoom e viste
Esporta dxf
Profilo stratigrafico
Analisi
Generale
Nome profilo
Profilo stratigrafico (1)
Mesh
Tipologia mesh
Quadrilateri
Passo
0.50
m
Modello
Indefinito lungo X
Carichi statici
Moltiplicatore Peso Proprio
0.00
Colonna di controllo (1)
(2)
(4)
(6)
(8)
Selezione: Selezionare gli elementi della stratigrafia per modificare le caratteristiche.
D:\LSR_2_ Nuovo Manuale\Analisi mono.lsr

Main window components
Buttons of the main window are listed below.

Data

- Clicking 

opens the editor of General Data.
- Clicking 

opens the editor of acceleration-time curves.
- Clicking 

opens the editor of decadence curves.

Model

- Clicking 

enables/disables the feature that allows you to select model elements.
- Clicking 

enables/disables the feature that allows you to add new points to the model or select existing ones.
- Clicking 

enables/disables the feature that allows you to add new line segments to the model or select existing ones.
- Clicking 

enables/disables the feature that allows you to break a line segment by adding a new intermediate point into the model.
- Clicking 

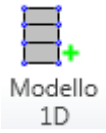
enables/disables the feature that allows you to add new layers to the model or select existing ones.
- Clicking 

enables/disables the feature that allows you to add new columns to the model or select existing ones.

- Clicking 

deletes the entire model.

Utilities

- Clicking 

automatically creates 1D model.
- Clicking 

lets you import model from a dxf/dwg file.

Calcolation

- Clicking 

enables/disables mesh generation.
- Clicking 

runs calculation.

Output

- Clicking 

opens plot window where you can view calculation results.
- Clicking 

opens the window where you can view calculation results for check points.
- Clicking 

generates calculation report.

Zoom e Viste

- Clicking 

lets you zoom the view.
- Clicking 

lets you export the model to a dxf file.

Secondary windows

Acceleration-time curves window

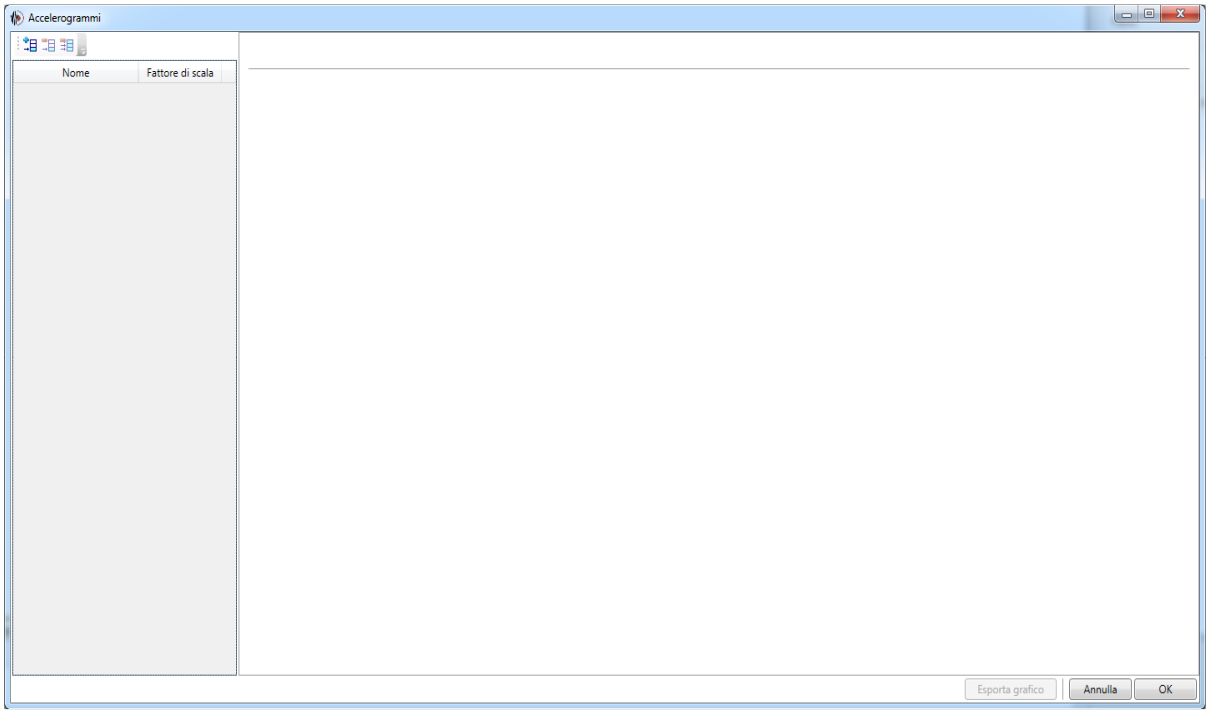


Fig. 2.2 – Acceleration-time curves window.

Buttons of the window are listed below.

Add a new acceleration-time curve.

Delete the selected acceleration-time curve.

Delete all acceleration-time curves.

1D Model window

By this window you can define properties of the stratigraphic column for 1D analysis.

You can open this window by clicking 

in main window.

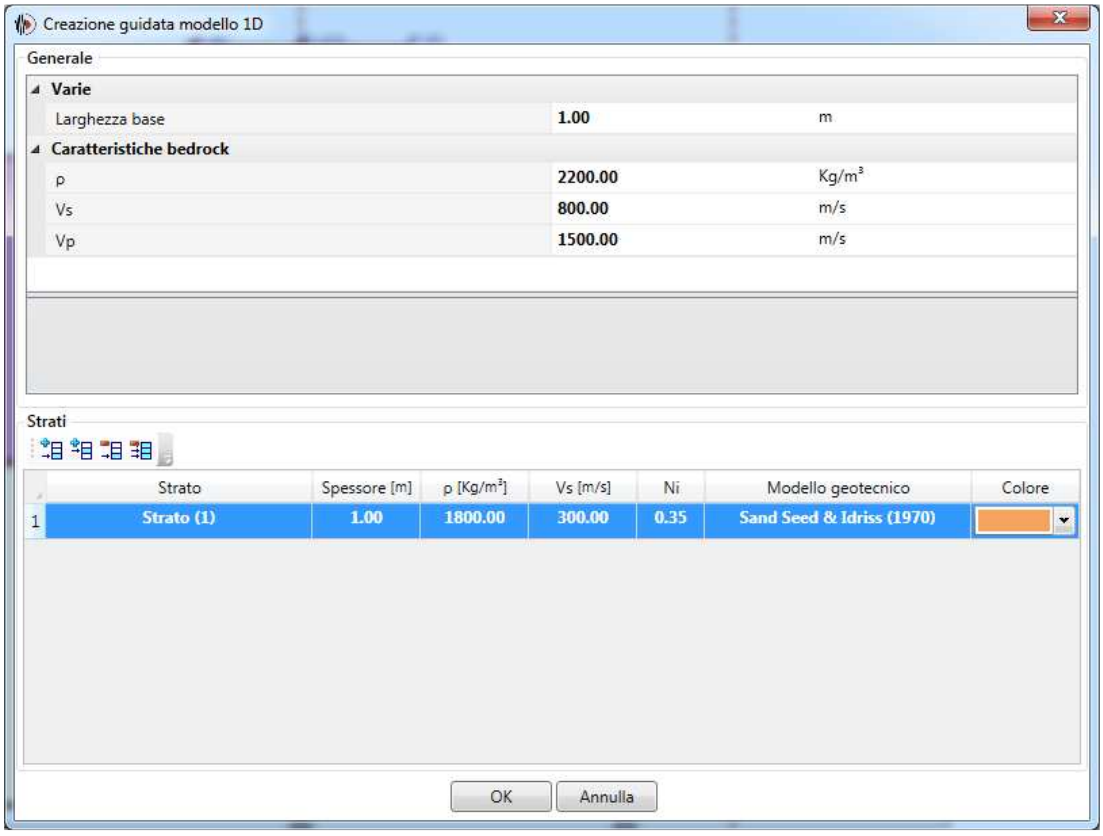
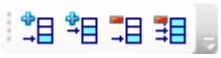


Fig. 2.3 – 1D model window.

Meaning of the controls in the window:

- **Generale**
  - **Larghezza base** – length of the base of stratigraphy
  - **Ro** – bedrock mass density
  - **Vs** – bedrock S-wave velocity
  - **Vp** – bedrock P-wave velocity
- Strati 
  -  Add a new layer
  -  Add a new layer before the selected one
  -  Delete the selected layer
  -  Delete all layers.

2.2 Plot – Viewing results

To the aim of viewing soil behaviour open the plot clicking 

in main window.
LSR2D lets you view results in the domain points and along check columns.

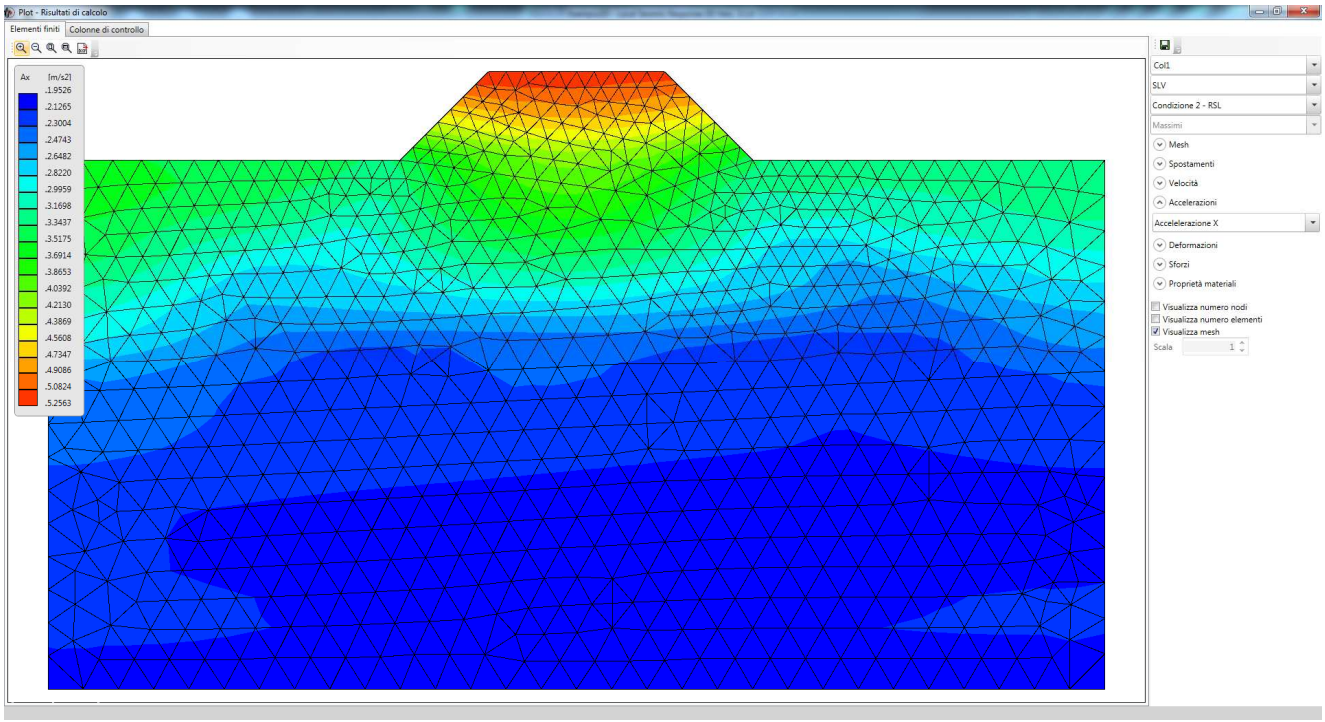
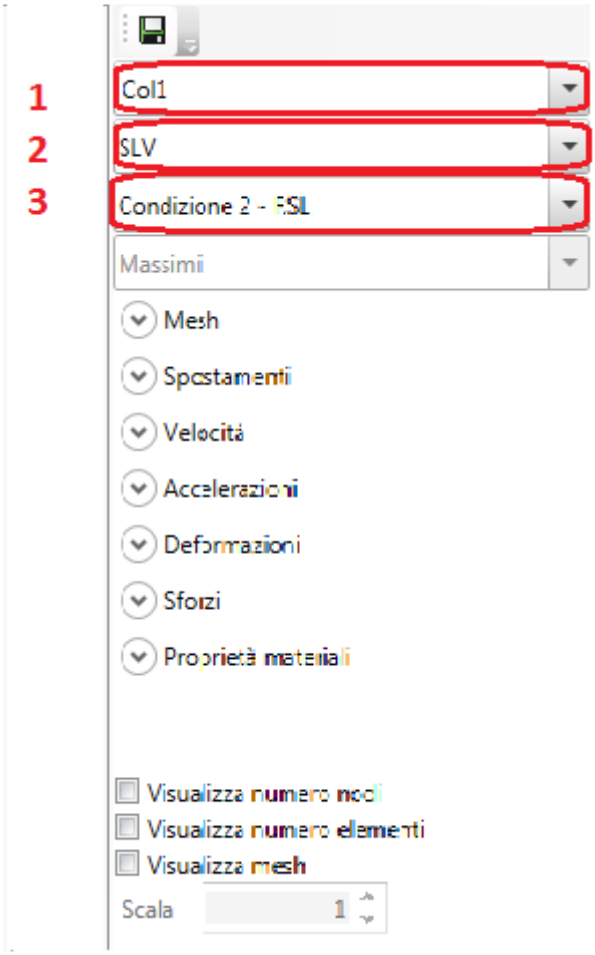


Fig. 2.4 –Plot – FEM calculation results.

To view results at domain points click ‘’Elementi finiti’’.

To view results along check columns click ‘Colonne di controllo’’.

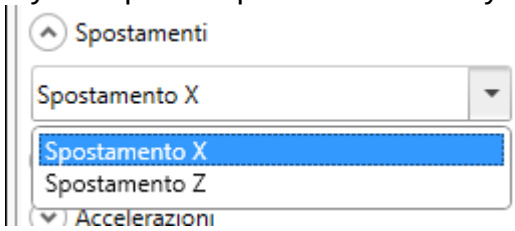


In the graphical user interface controls showed in the previous figure, you find:

-  lets you export to a .txt or .jpeg file the viewed graph related to the selected check column.

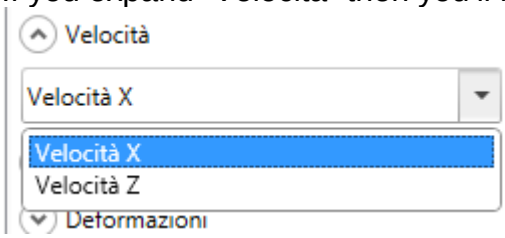
1. Combo box menu lets you select the check column for which see results.
2. Combo box menu lets you select the limit state for which see results.
3. Combo box menu lets you select the load case for which see results.

- If you expand “Spostamenti” then you’ll find the combo box menu



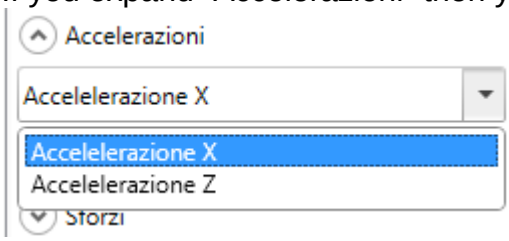
by which you can view results in terms of displacements in the domain points or along the selected check column.

- If you expand “Velocità” then you’ll find the combo box menu



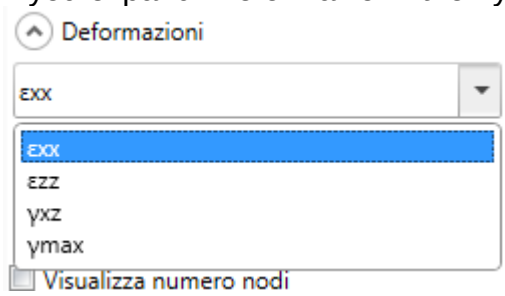
by which you can view results in terms of velocities in the domain points or along the selected check column.

- If you expand “Accelerazioni” then you’ll find the combo box menu



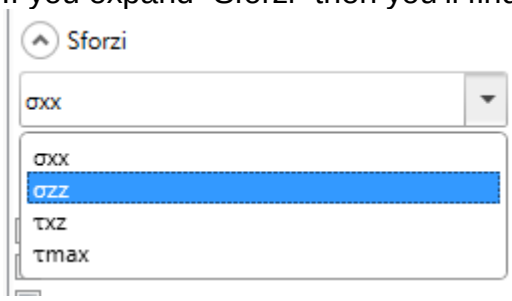
by which you can view results in terms of accelerations in the domain points or along the selected check column.

- If you expand “Deformazioni” then you’ll find the combo box menu



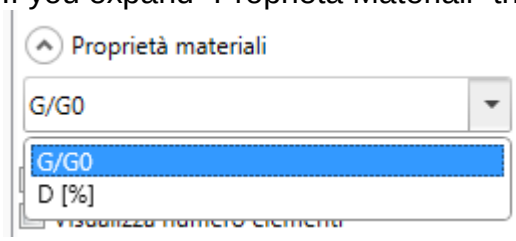
. by which you can view results in terms of deformations in the domain points or along the selected check column.

- If you expand “Sforzi” then you’ll find the combo box menu



by which you can view results in terms of tensions in the domain points or along the selected check column.

- If you expand “Proprietà Materiali” then you’ll find the combo box menu



by which you can view material properties in the domain points or along the selected check column.

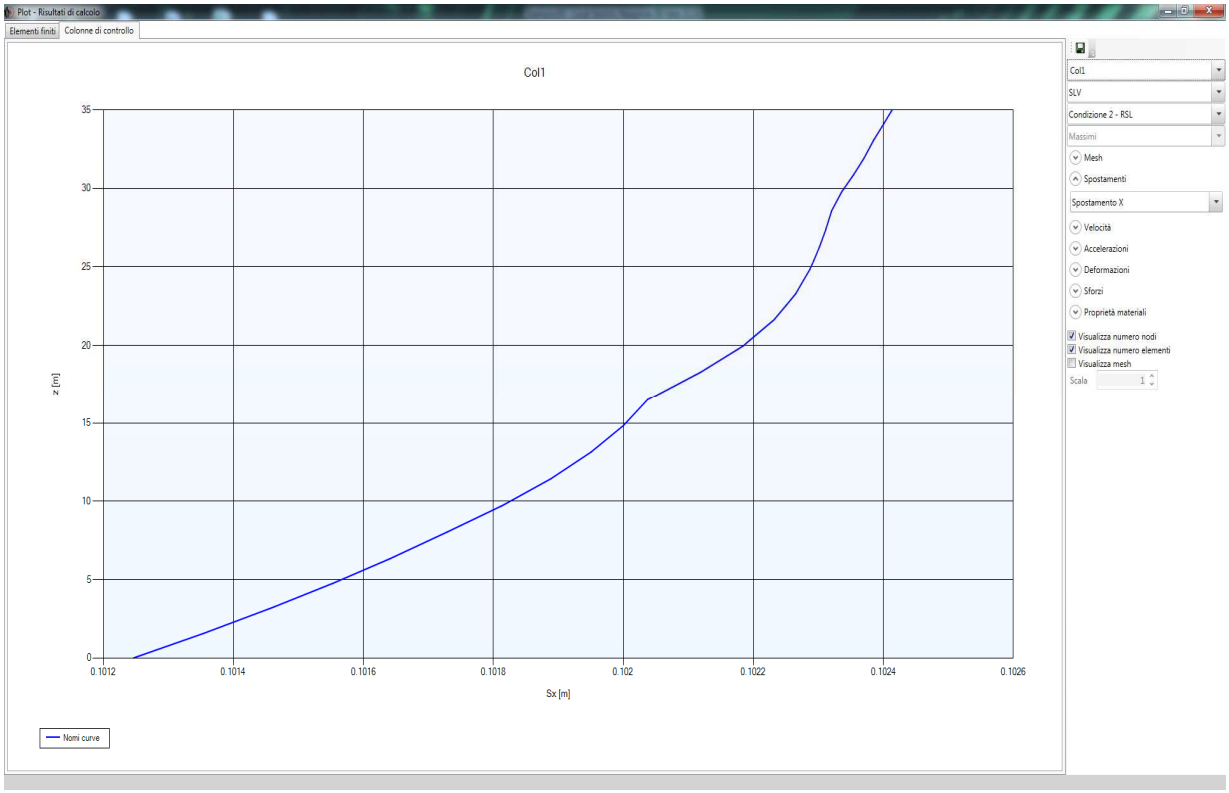


Fig. 2.5 –Plot – Check column results.

2.3 Check points results

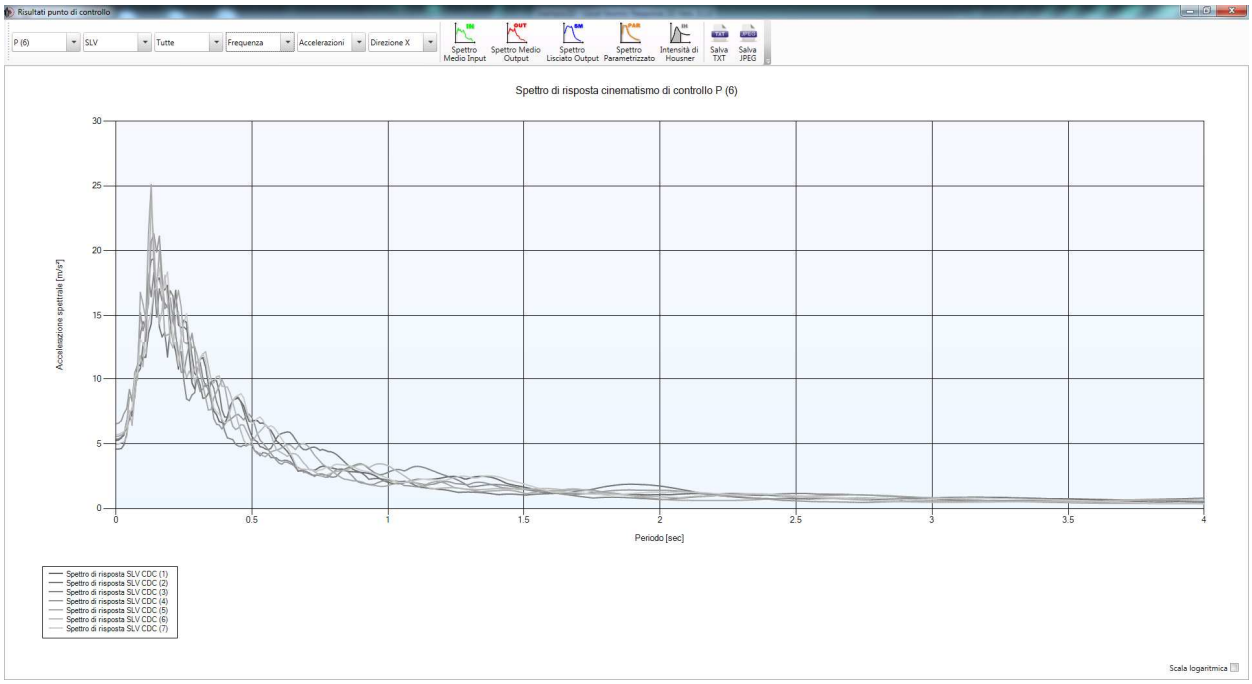
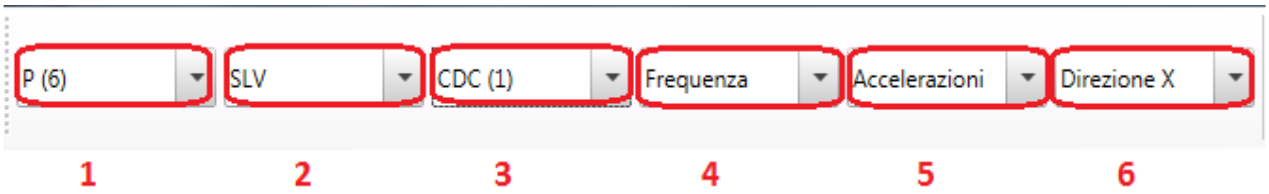
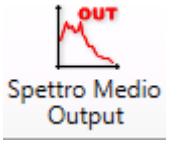


Fig. 2.6 – check point results window.

Controls and buttons of the window are listed below.

- 

  1. Combo box menu lets you select the check point for which see results.
  2. Combo box menu lets you select the limit state for which see results.
  3. Combo box menu lets you select the load case for which see results.
  4. Combo box menu lets you choose between time or frequency.
  5. Combo box menu lets you choose between displacement, velocity, acceleration.
  6. Combo box menu lets you choose between direction X (horizontal) and Z (vertical).
- 

shows average input spectrum (available only with frequencies).
- 

shows average output spectrum (available only with frequencies).
- 

shows smoothed output spectrum (available only with frequencies).
- 

shows parametrized spectrum (available only with frequencies).
- 

calculates Housner intensity (available only with frequencies).

- 

saves graph to a .txt file.

- 

saves graph to a .jpeg file.

3. Input

3.1 Input stages

Software main window is showed in figure 3.1. From this you can reach all the windows to define your model.
Modeling consists of the following steps:

- Defining general data;
- Uploading acceleration-time curves and scaling them by peak ground acceleration;
- Defining decadence modeling for soil shear modulus and viscous damping;
- Defining soil layers;
- Defining check points on ground surface;
- Defining check columns;
- Mesh generation;
- Defining load cases.

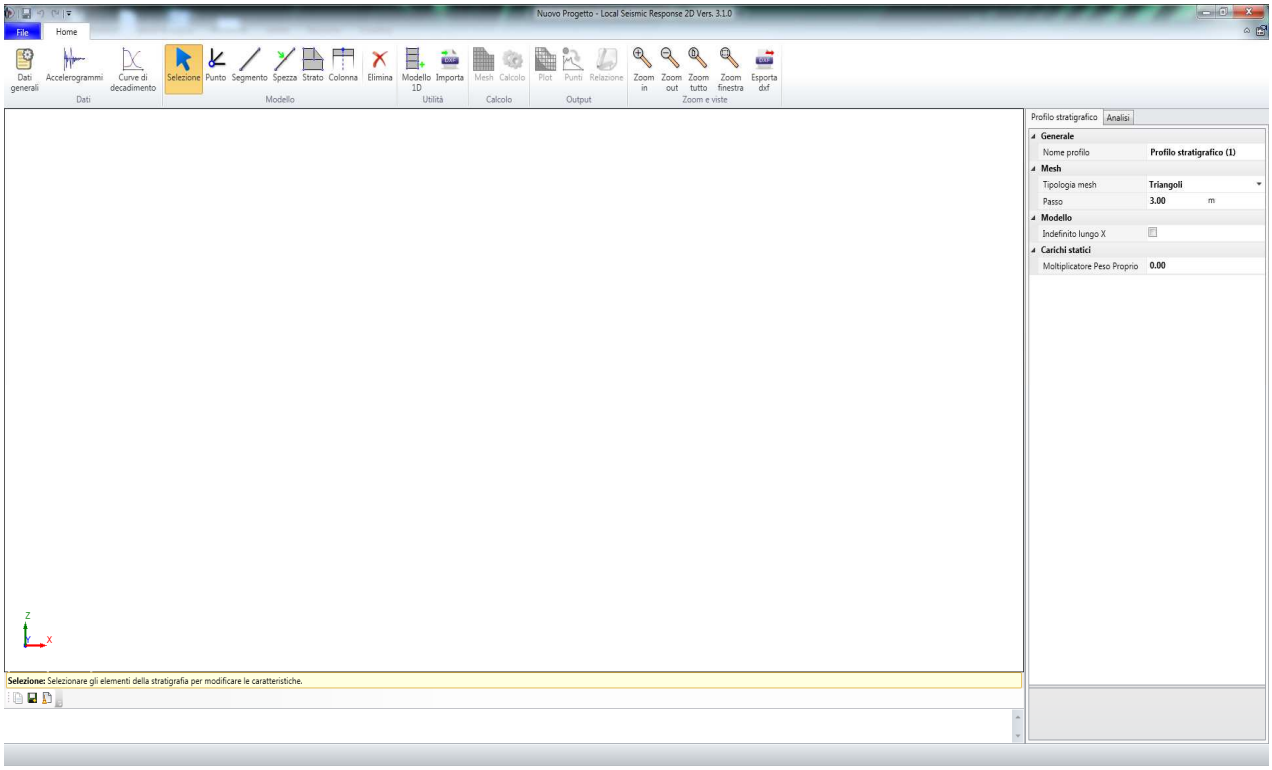


Fig. 3.1 *–LSR2D main window.*

3.2 Entering data – General data

You can open the window showed in ***fig. 3.2*** by clicking 

in the main window***.***

By the item **"Dati generali calcolo"** you can edit:

- number of threads ("Numero di threads");
- enabling/disabling setting material properties to zero for each load case ("Azzera caratteristiche materiali");
- relative error on shear modulus decadence ("Errore relativo sul decadimento G");
- relative error on damping ratio decadence ("Errore relativo sul decadimento D");
- choice between plane stress and plane strain ("Modello piano");
- effective deformation ratio ("Rapporto deformazione effettiva");
- maximum number of iterations for eigenvalue extraction ("Numero massimo di iterazioni");
- enabling/disabling FE model view ("Visualizza modello FEM");
- enabling/disabling selection of natural modes ("Seleziona modi naturali");
- minimum output spectrum period ("Periodo minimo spettro");
- maximum output spectrum period ("Periodo massimo spettro");
- output spectrum period step ("Periodo passo");
- damping ratio ratio expressed as percentage for the calculation of the response spectrum ("Smorzamento riferimento");

- Users having the license of QUAD4-M (available by specific vendors and not by Stacec) can run QUAD4-M solver directly from LSR_2D as an alternative to the internal solver. To this aim you have to :
    - A) check the option ("Attiva QUAD4-M");
    - B) define bedrock S-wave velocity ("Vs bedrock");
    - C) define bedrock P-wave velocity ("Vp bedrock");
    - D) define bedrock density ("Densità del bedrock");

- Coefficient to determine effective shear deformation from maximum one;

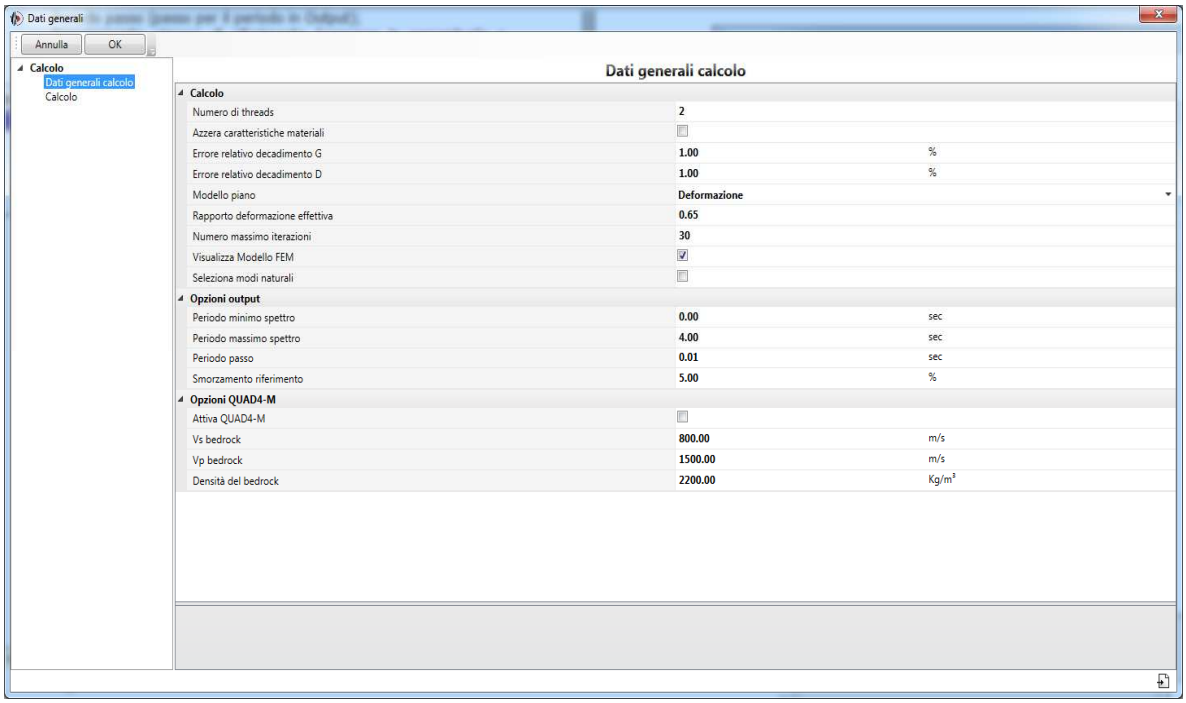


Fig. 3.2 – *General data.*

By the item **"calcolo"** you can edit:

- Eigenvalue extraction tolerance ("Tolleranza calcolo autovalori");
- Eigenvalue extraction maximum iteration number ("Numero massimo iterazione autovalori");
-

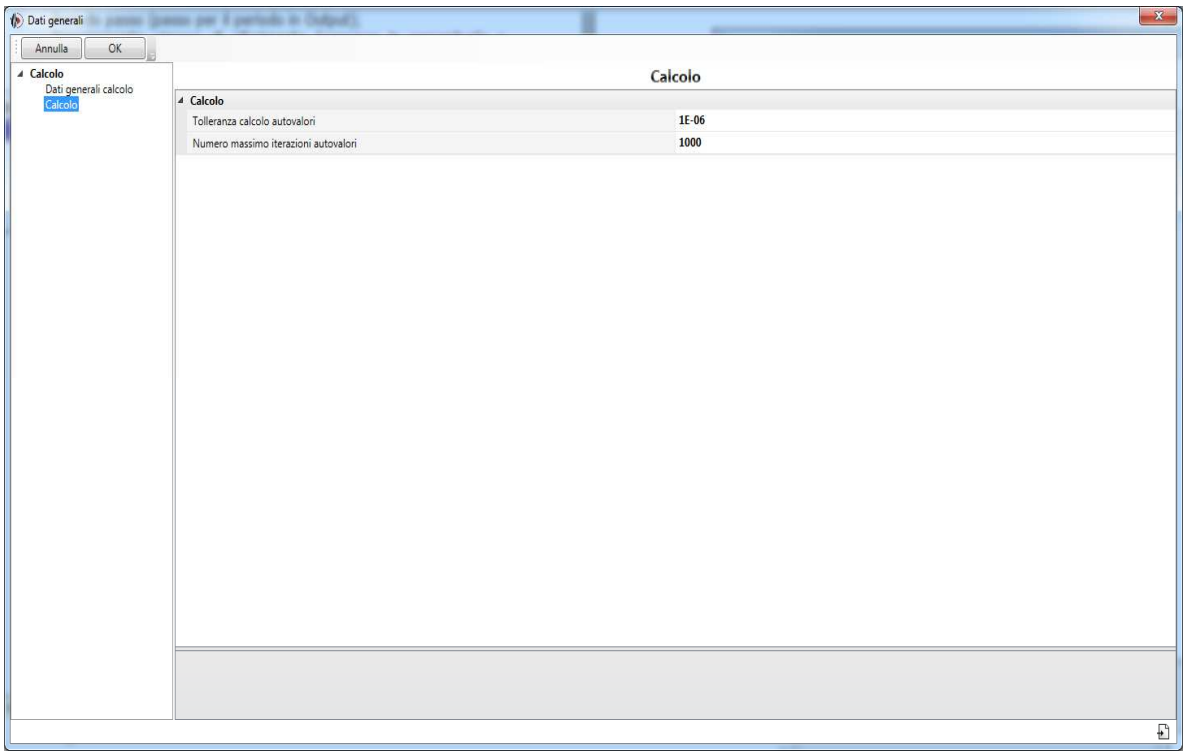


Fig. 3.3 – *General data - Calculation.*

3.3 Input – Acceleration-Time curves

By clicking 

in the main window, you can open the windiow showed in ***fig. 3.4.***

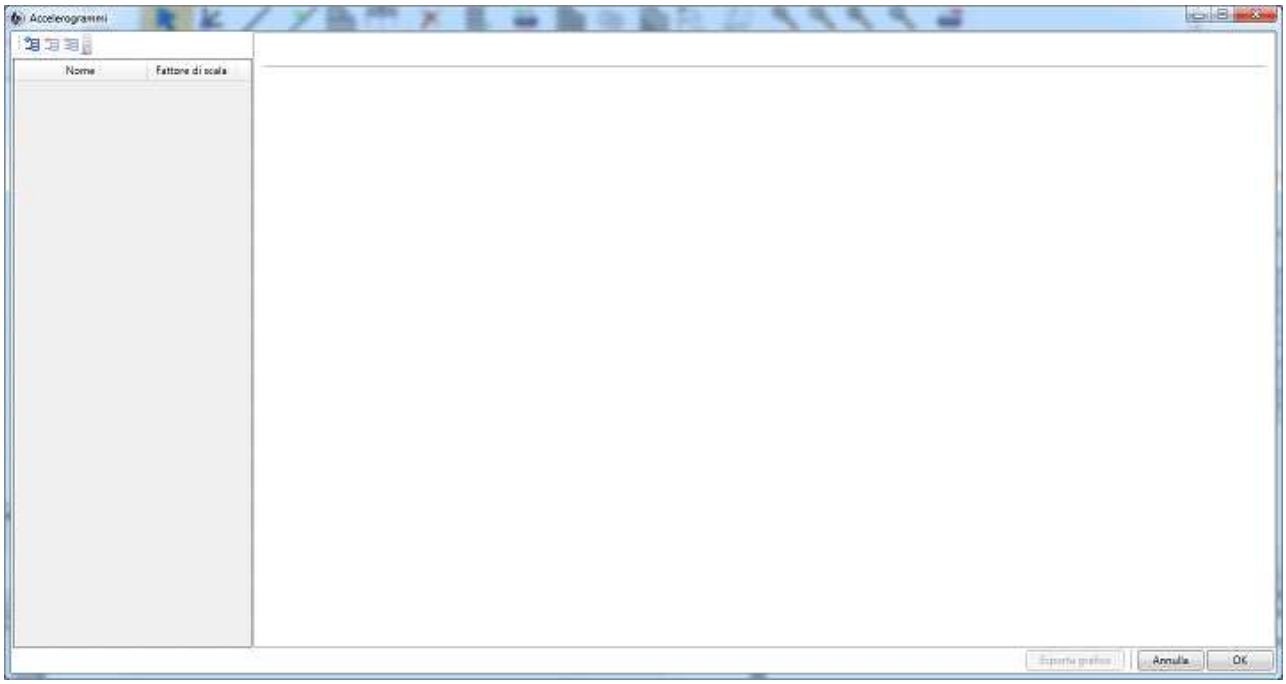


Fig. 3.4 – *Acceleration-time curves.*

The window lets you load acceleration-time curves from *.txt file.

Text format in the file should be:

0.00 5.70E-04
0.01 1.55E-03
0.02 2.26E-03
……

First column contains time instants and secondo ne accelerations.
By clicking you have to define file path, then you specify measurement units and the value of peak ground acceleration which scale the curve (see Fig. 3.5).

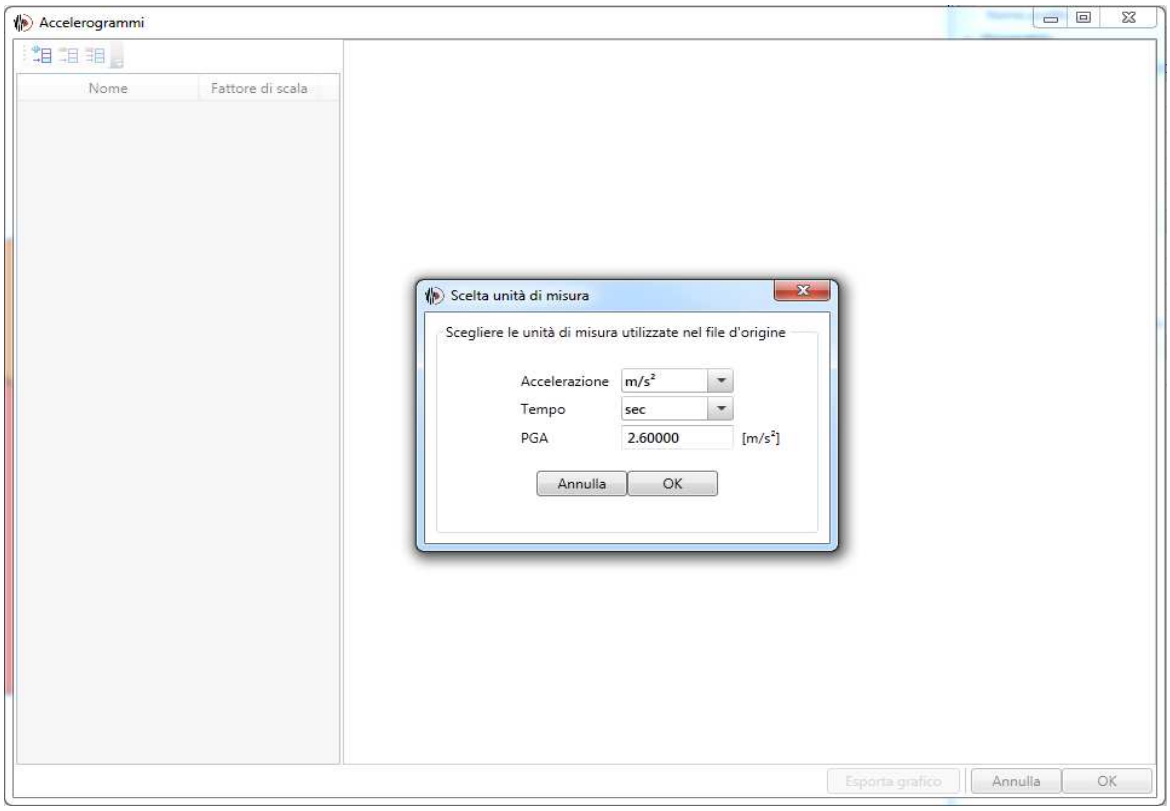


Fig. 3.5 – *Add an acceleration time curve.*

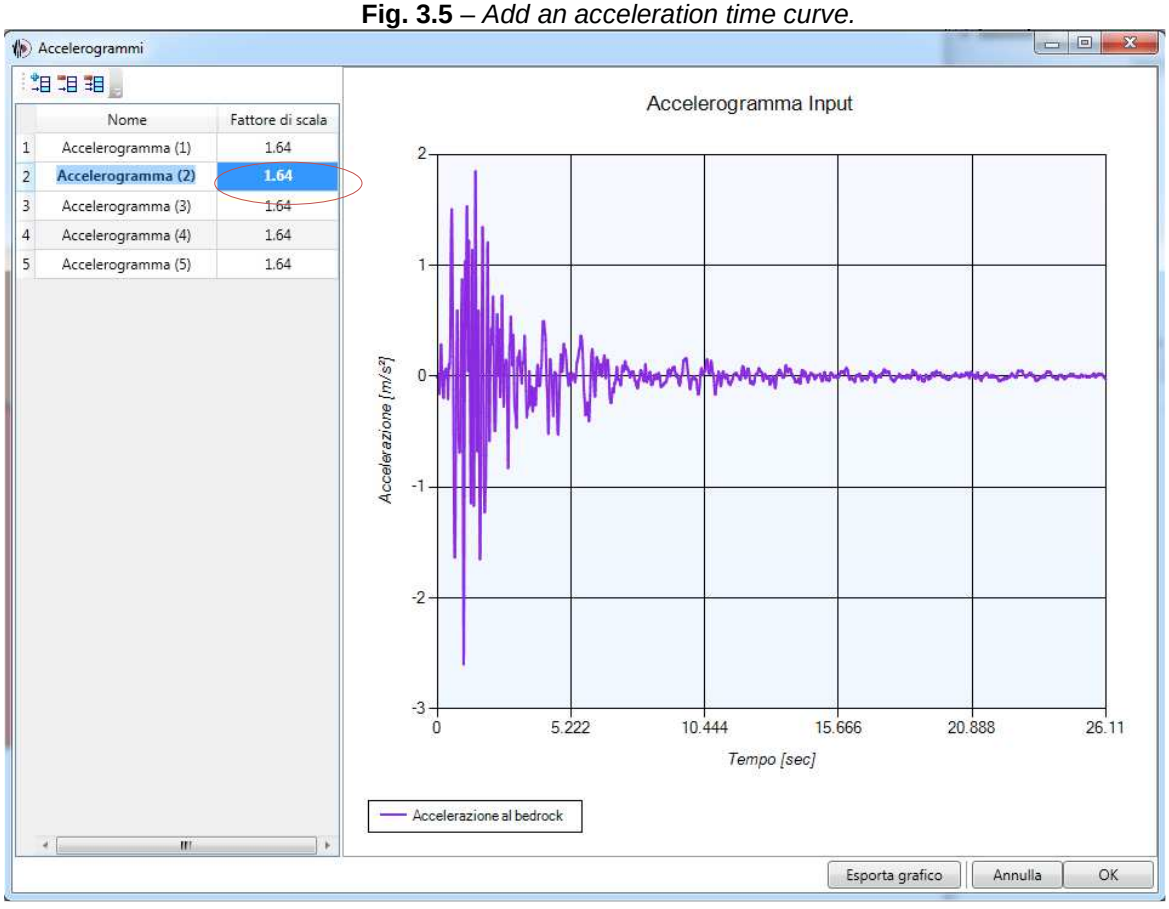


Fig. 3.6 – *Viewing an acceleration-time curve.*

N.B. Scale factor (circled in red in Fig. 3.6) can be mean as tha ration between user PGA used when adding the acceleration-time curve and the acceleration peak of the record added. This value can be changed by the user in every moment.

Holding “Ctrl” key you can select more than one acceleration-time curve (see Fig. 3.7) and scale all of them.

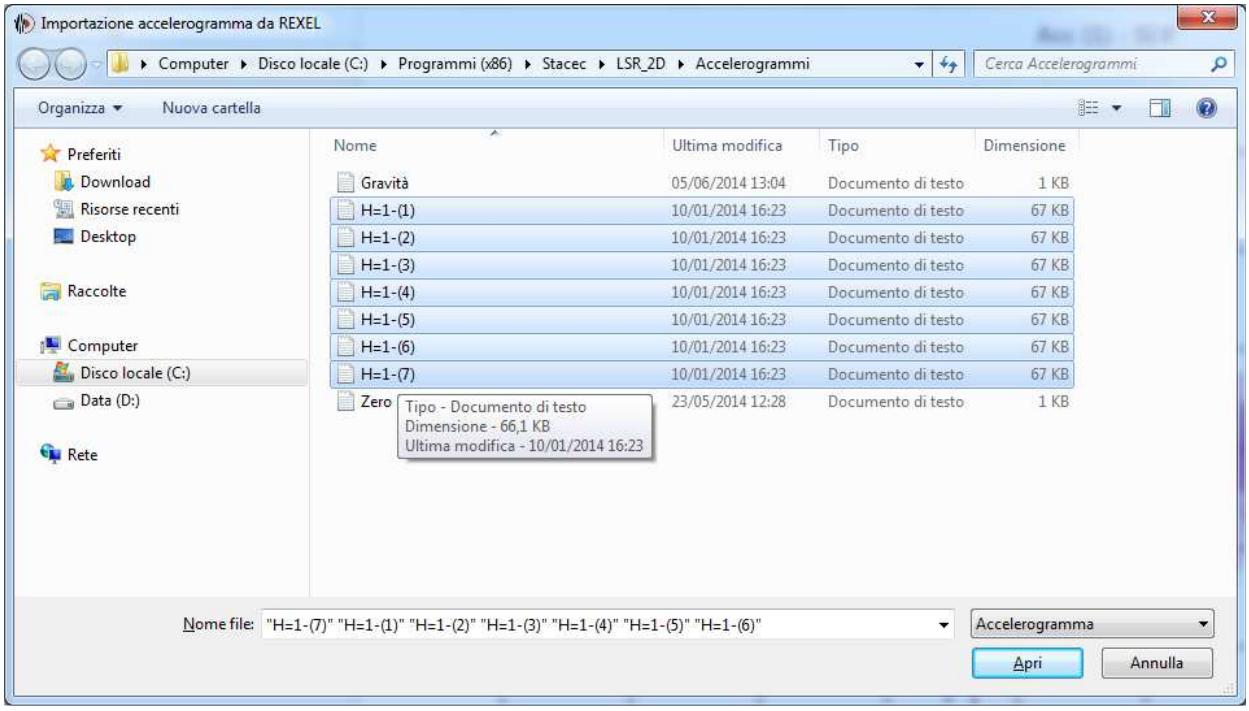


Fig. 3.7 – *Multiple selection of acceleration-time curves when uploading.*

3.4 Input - Decadence curves

Linear equivalent model, on which calculation is based, takes into account γ soil shear deformation-dependent variation of shear modulus G and viscous damping ratio D. These properties must be assigned to each layer on the basis of the nature and appropriate tests.

Clicking 

in the main window you open this editor which is showed in ***fig. 3.8.***

You can assign to each layer a couple of decadence curve among the ones loaded in the editor; see paragraph on the definition of layers.

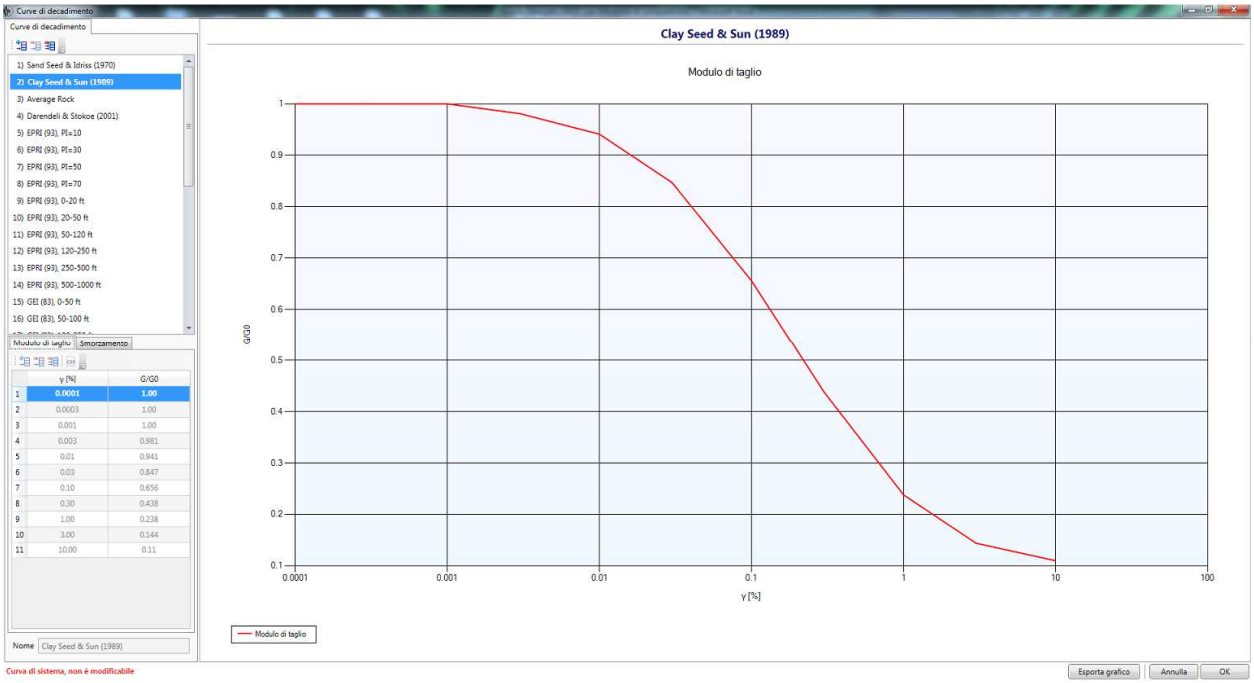
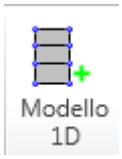


Fig. 3.8 – decadence curves editor.

Default curves are based on material models available in literature. You can add other curves to the editor, by clicking . These curves can be defined manually or by loading a .csv file.

3.5 Defining model – Stratigraphy for 1D analysis

Clicking 

in main window, you open the wizard **Creazione guidata modello 1D** showed in ***fig 2.8.***

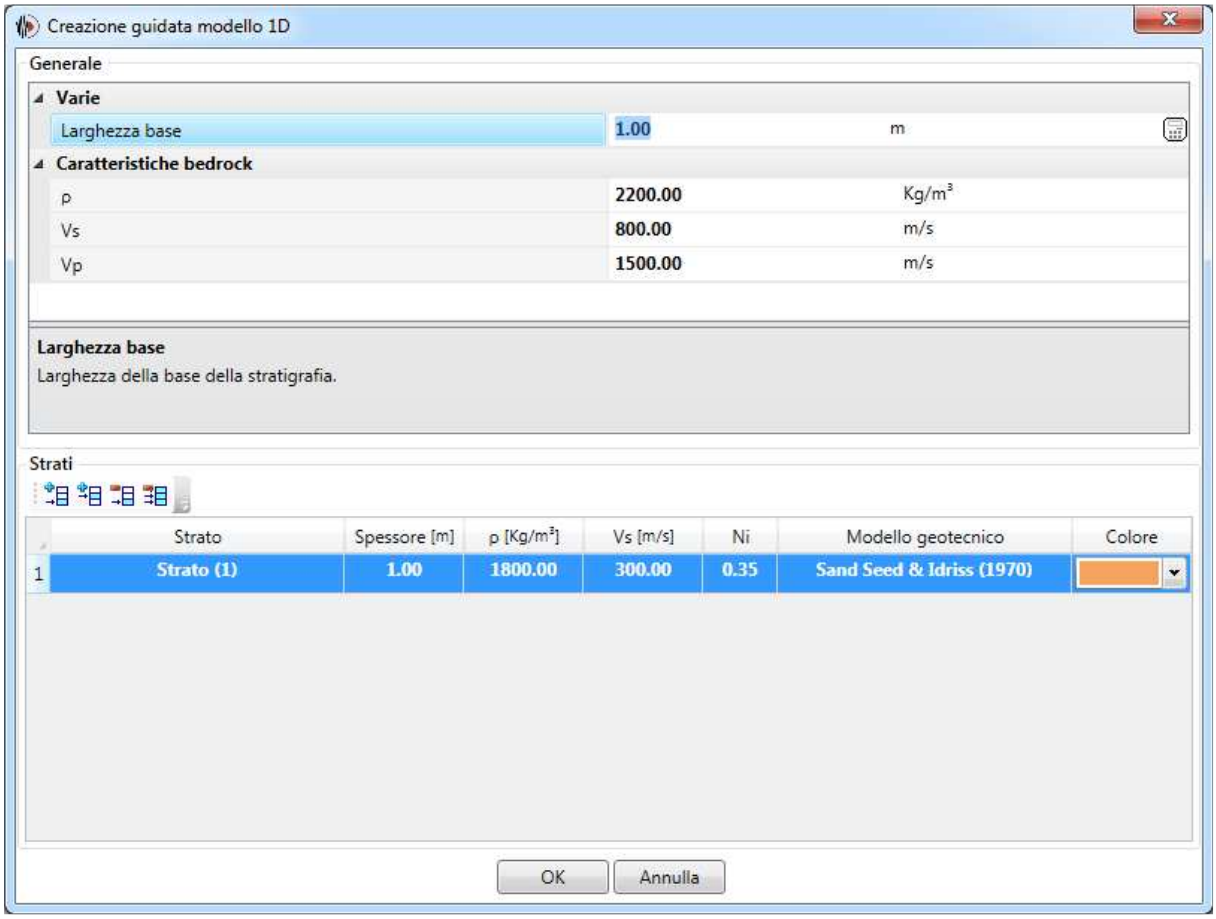


Fig. 3.9 – *Creazione guidata modello 1D.*

In this window you can edit:

- Length of the base of the column;
- rho – density of the bedrock;
- vs – bedrock s waves velocity;
- vp – bedrock p waves velocity.

You can also edit number and properties of each layer.

- Name (“Strato”);
- Thickness (“Spessore”);
- Density (rho);
- S wave velocity (Vs);
- Poisson coefficient (Ni);
- Decadence model.
- color.

By clicking  you add a layer.

3.6 Defining model – Startigraphy for a 2D analysis

2D stratigraphy model defining consists of points and lines which connect these points and define the boundaries of the layers.

Basic Input

Point definition - Click , then click in the graphical area: a point will be added. By clicking the point you select it and can edit its properties:

- Name;
- Specifying if it's a check point;
- X and Z coordinates;
- X and Z cinematics;
- Mesh size;
- Fx and Fz static loads.

Line segment definition - Click , then click a sequence of points in the graphical area or a on points already defined to add one or more line segments.

Layer definition - Click , then click on the vertices or in the inside of a mono-connected polygon or box-select one or more mono- connected polygons to add one or more layers. Select one or more layers to edit layer properties:

- Name;
- Description;
- Color;
- Decadence curve model for stiffness and damping;
- r – density of material;
- Vs – S waves velocity;
- Ni – Poisson coefficient;
- Gini – initial G module;
- Dini – initial viscous damping ratio;

Fig. 3.10 – *Single layer data.*

You can also import a stratigraphyfrom a dxf or dwg file. Click 

to open the import window,

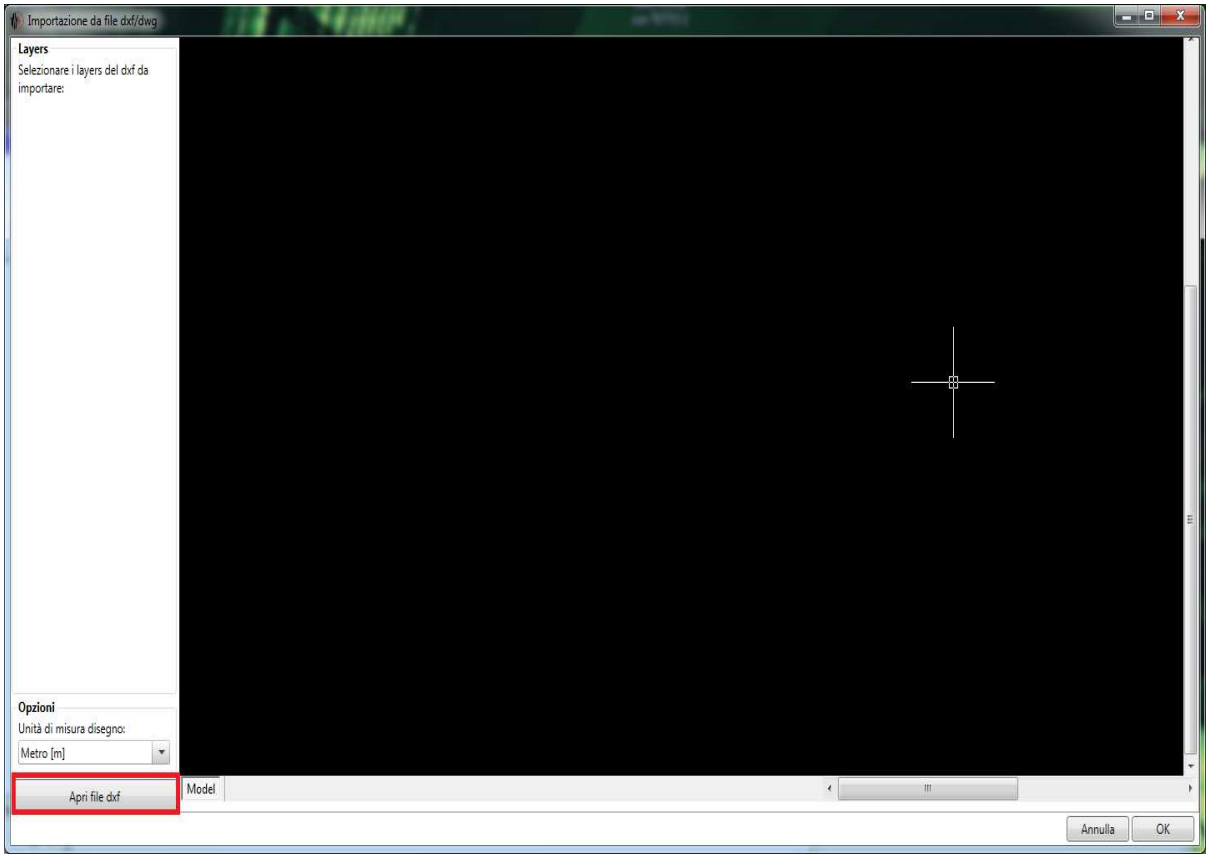


Fig. 3.11 – import from dxf/dwg.

Select measurement units of the drawing, click “Apri File dxf” and select the file to import.

Then you have to define and assign layer properties.

3.7 Defining boundary conditions

Select boundary and assign the restraint “**Vincolato Z”** (see figg. 3.12 and 3.13).

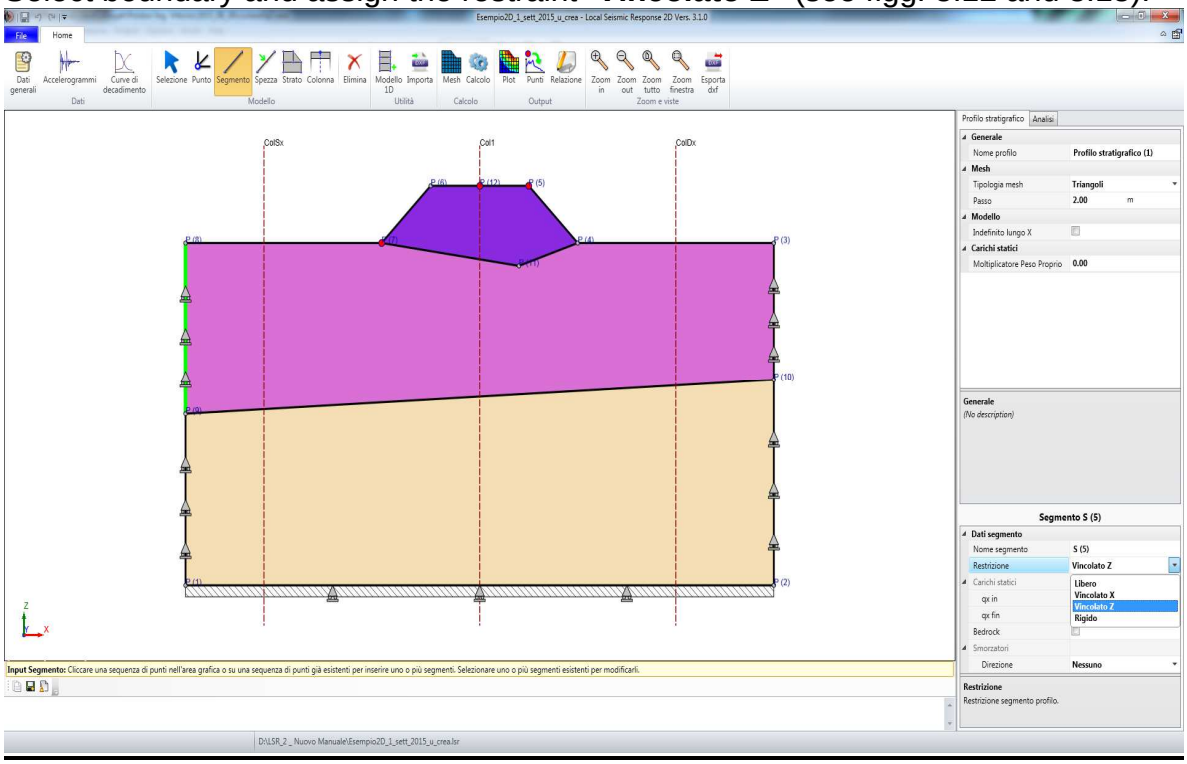


Fig. 3.12 – *Boundary conditions*

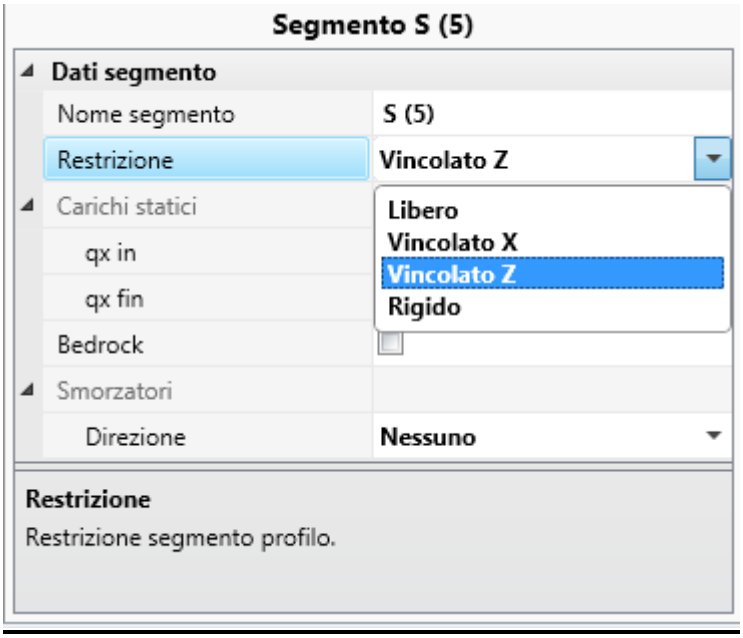


Fig. 3.13 – *Boundary conditions*

Definizione del BedRock

Select bottom boundaries, check “Bedrock” and customize parameters.

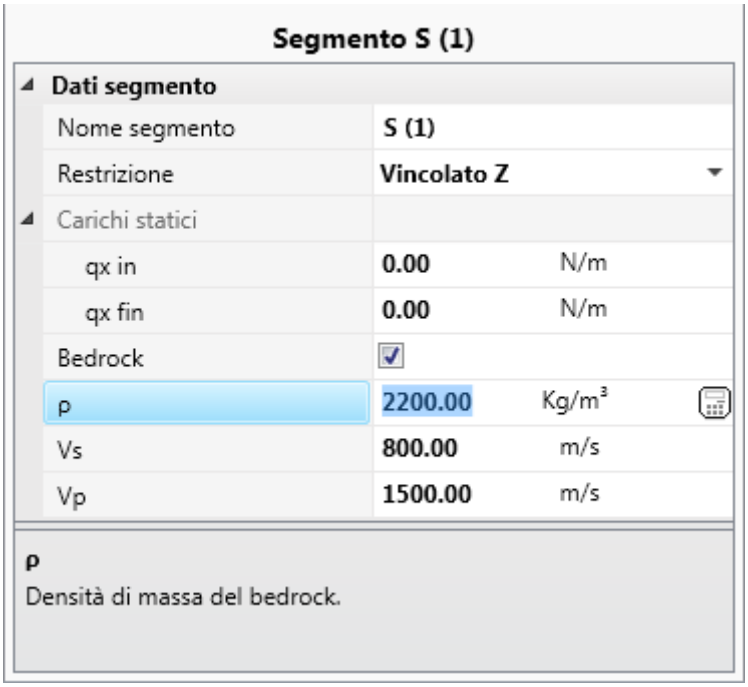
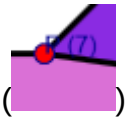


Fig. 3.14 –Bedrock data.

3.8 Defining check points

These points are the ones for which you have frequency domain results.

Click 

and select the point. Check the “Punto di controllo” flag property (Fig.3.15). I twill be marked as red (

).

You can define a local mesh refinement by the local mesh size (“Passo”).

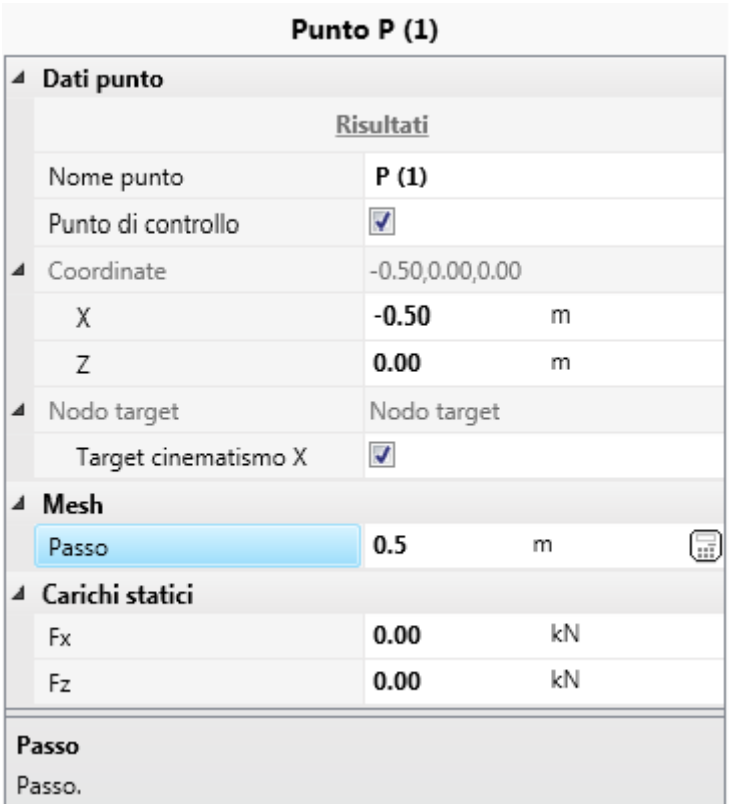


Fig. 3.15 – Check point properties.

3.9 Defining check columns

Displacements, velocities, accelerations, deformations and stresses results will be available for check columns.

Click , then click on an active point of the stratigraphy to add a new column. Clicking on a column you can edit its name and position.

Fig. 3.16 – Check column data.

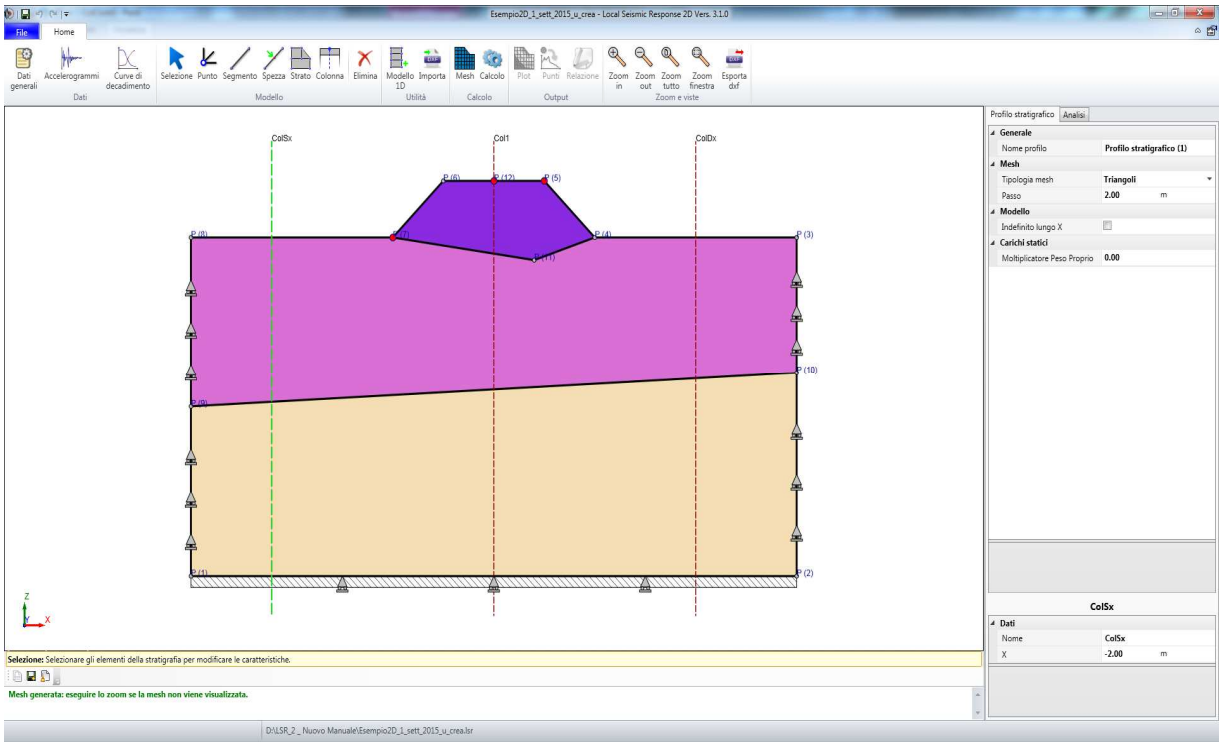


Fig. 3.17 – Check column definition.

3.10 Defining load cases

You can define one or more analyses (“Analisi”) and for each of these analyses you can define one or more load cases (“Condizioni di carico”). A load case is characterized by an X (horizontal) and a Z (vertical) acceleration-time curve.

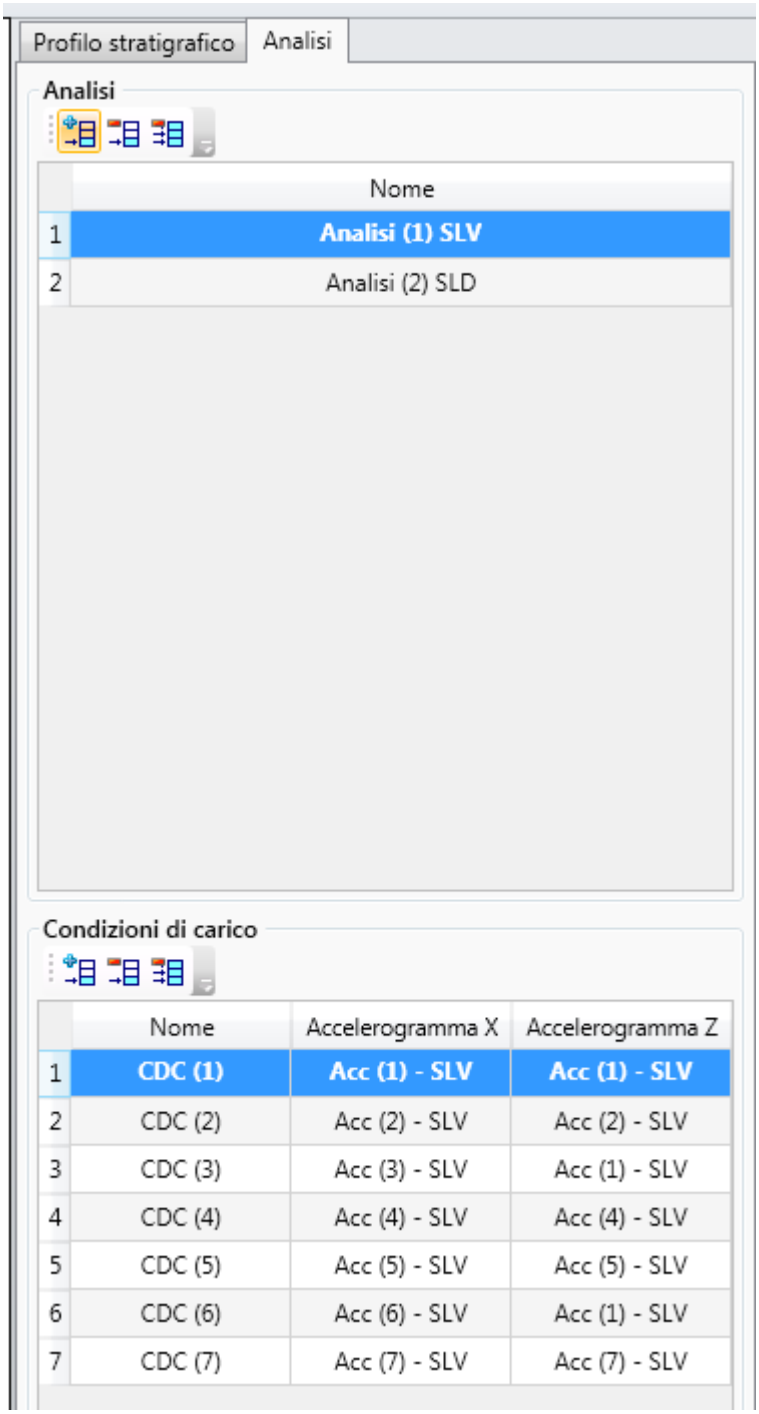


Fig. 3.18 – Analyses, load cases.

Click  respectively in the "Analisi" frame or in the "Condizioni di carico" frame to add an analysis or a load case (remember that for each analysis you should add 7 load cases as stated by Italian building code NTC 2008).

4. Mesh and Calculation

4.1 Mesh generation

You can customize general mesh parameters (fig 4.1)

- Mesh type (quadrilateral/triangular)
- Mesh size ("passo").

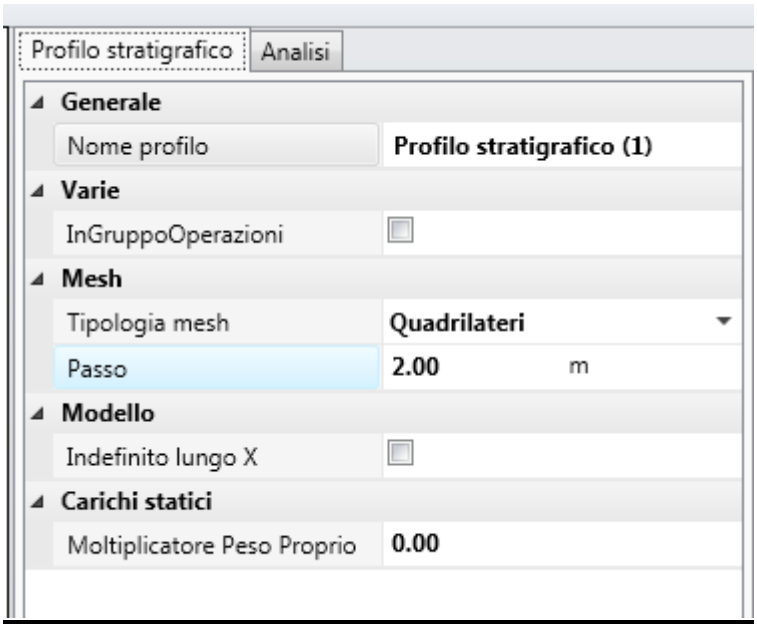


Fig. 4.1 – Mesh parameters.

And you can customize local check point mesh size to get a more accurate solution in the neighbourhood of the point (see paragraph 3.8).

Click 

or 

To generate and view mesh (see fig. 4.2).

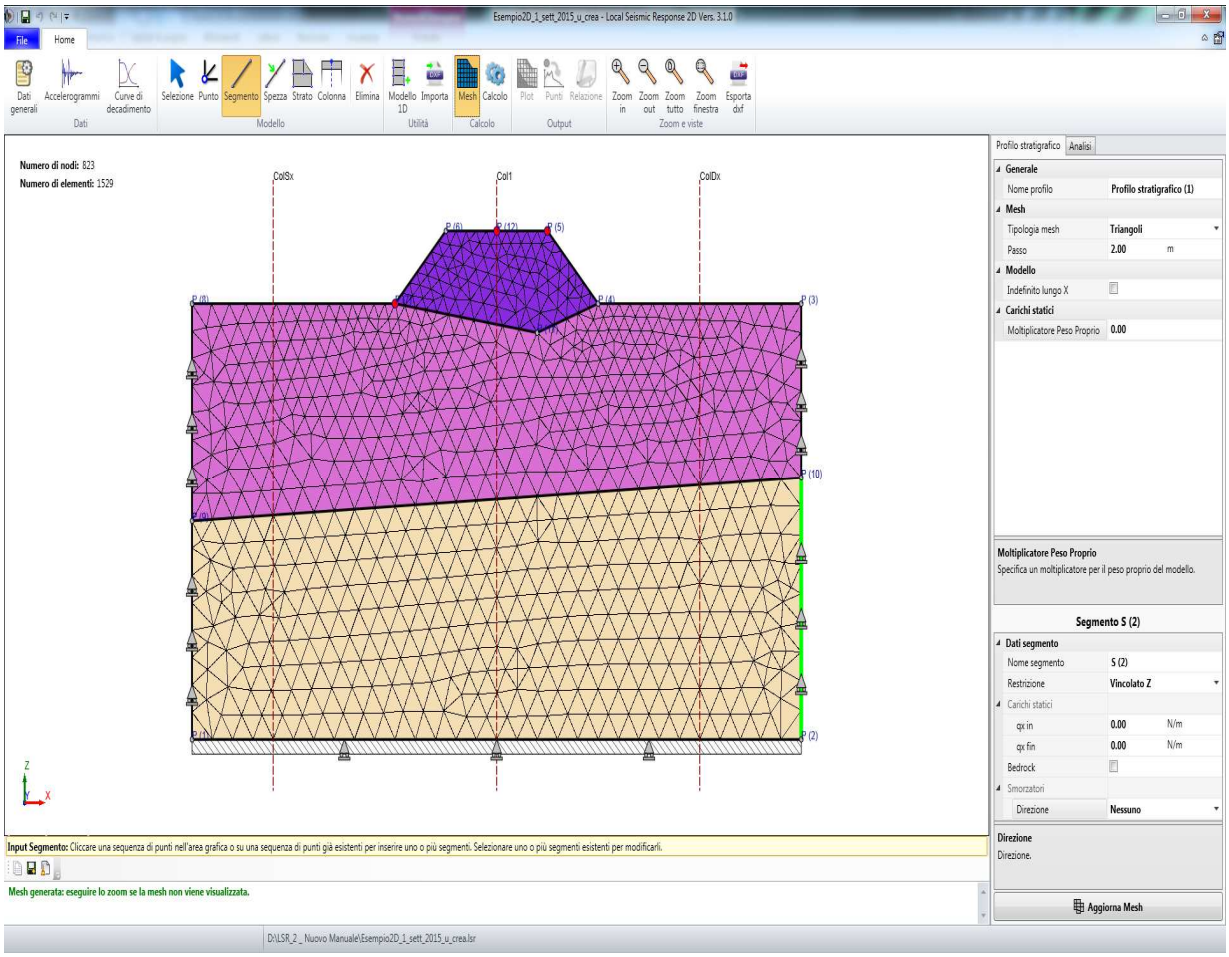


Fig. 4.2 –Mesh generation.

4.2 Calculation

Once the input is defined you can run processing by clicking 

; dialog window showed in fig 4.3 open.

During processing you will see in the dialog window:

- Finite element model (top of the window);
- Bedrock response spectrum for the load case being processed (curve ——);
- Check point response spectrum for the load case being processed (curve —— visible at the end of each load case calculation);
- Elapsed times and informations on the status of the model at each step;
- Total and partial advancing stage by progress bar.

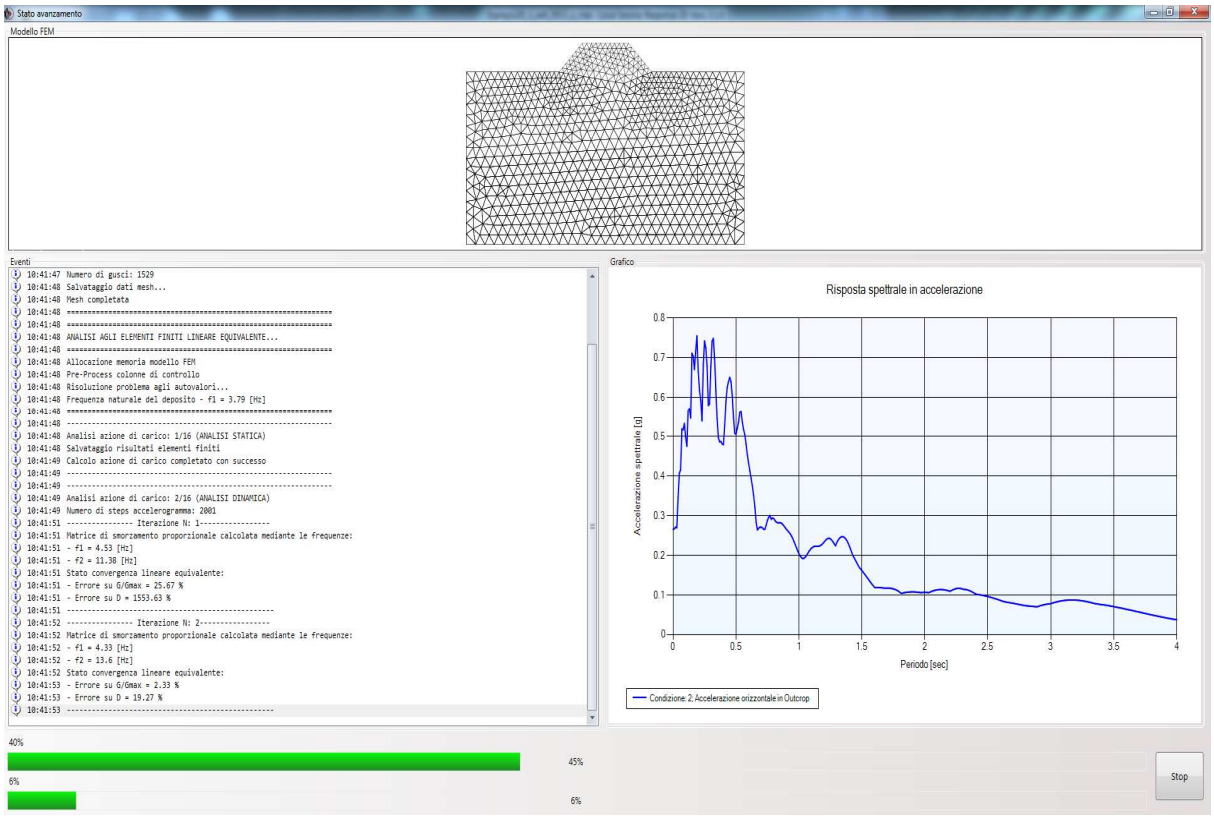


Fig. 4.3 – Progress of the FEM processing.

5. OutPut

5.1 Viewing point results

Clicking Plot opens window showed in fig. 5.1.
There you can view soil response in each mesh point in terms of displacement, velocity, acceleration, deformation and stress.

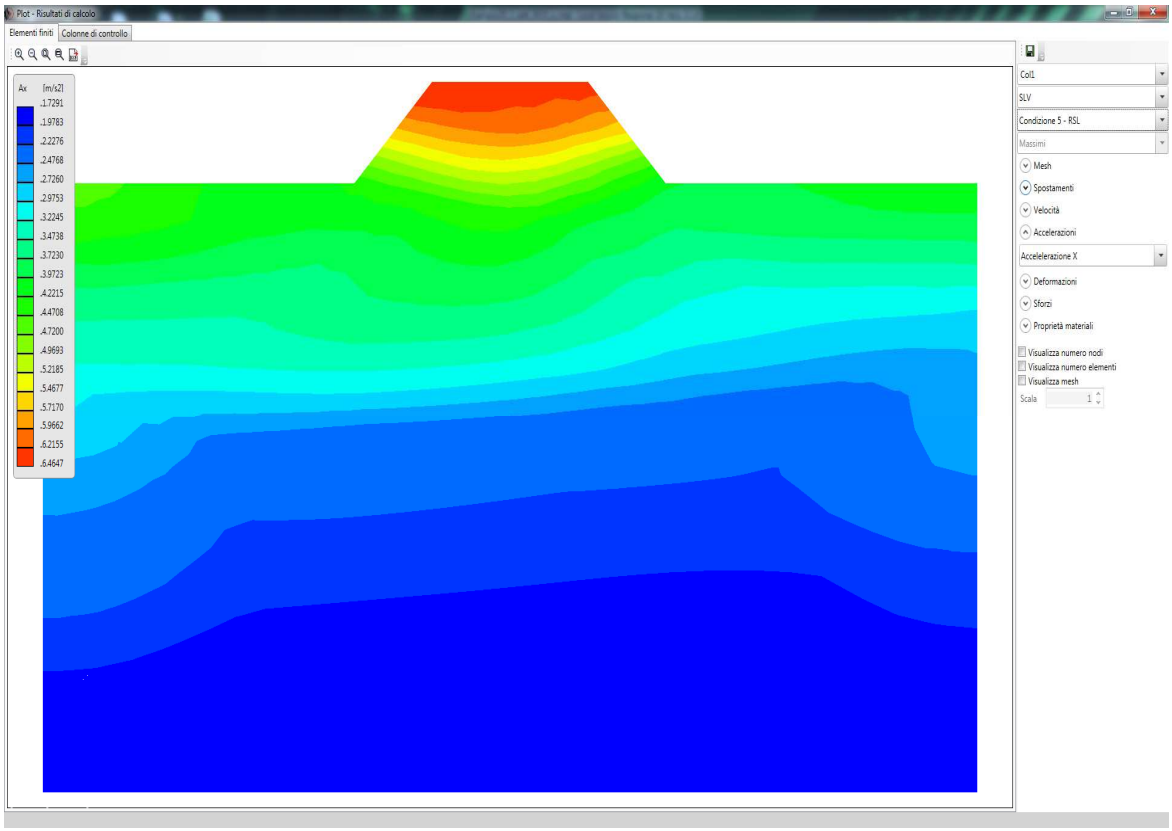


Fig. 5.1 – Viewing results.

Clicking tab page "**Colonne di controllo"** you can view check column results; results can be saved to .txt or .jpeg file (fig 5.2).

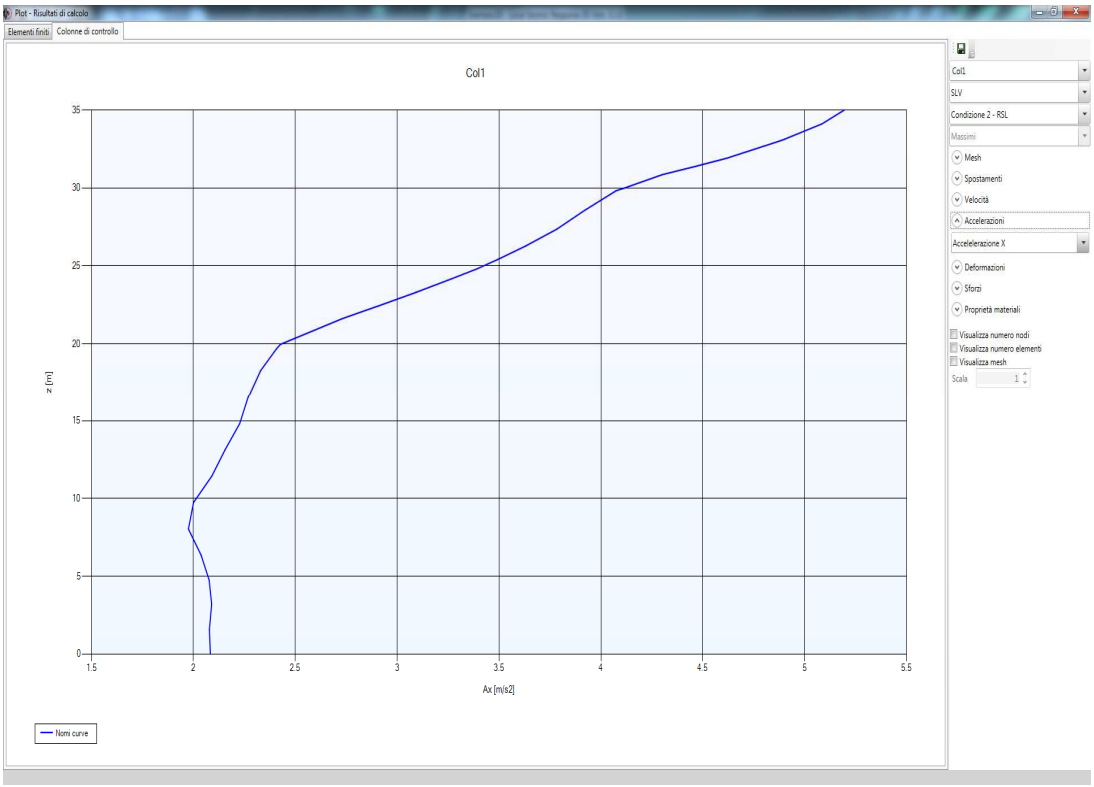


Fig. 5.2 – Check column results.

5.2 Viewing time and frequency results

Check point results

To open check point results window ("**Risultati punto di controllo**"), from main window click 

or select a point, right-click and select "Risultati".

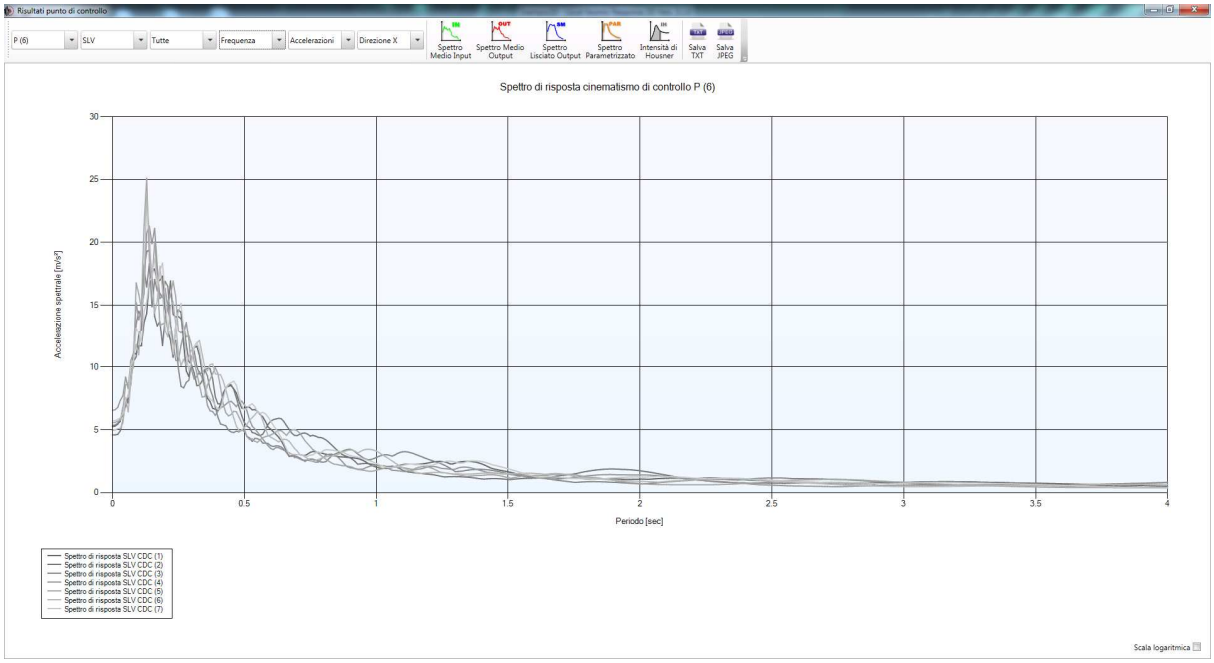


Fig. 4.2 – Check point results window.

In the window you find:

- **(1)** Combo box to select check point (if the point hasn't been selected in the main window)
- **(2)** Combo box to select analysis
- **(3)** Combo box to select load case
- **(4)** Combo box to choose between time or frequency domain
- **(5)** Combo box to choose between displacement, velocity or acceleration
- **(6)** Combo box to choose between X or Z direction.

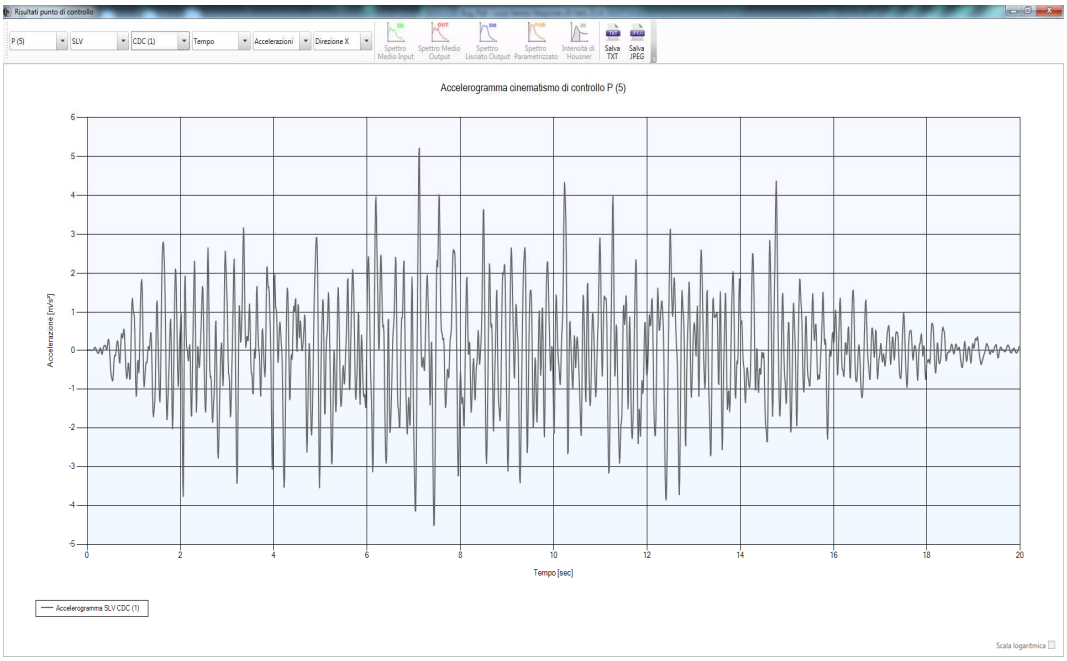


Fig. 4.3 – Check point acceleration-time curve.

Once you defined these options (as in fig. 4.4):

(4) Click on the save icon (**"Esporta**") to get currently plot graph points to .txt or .jpeg file

(5) In **"Scelta unità di misura"** window confirm [sec.] and [m/s^2] clicking **"ok"**.

(6) Select type of spectrum.

(7) Name it and click **"Salva"** to save it

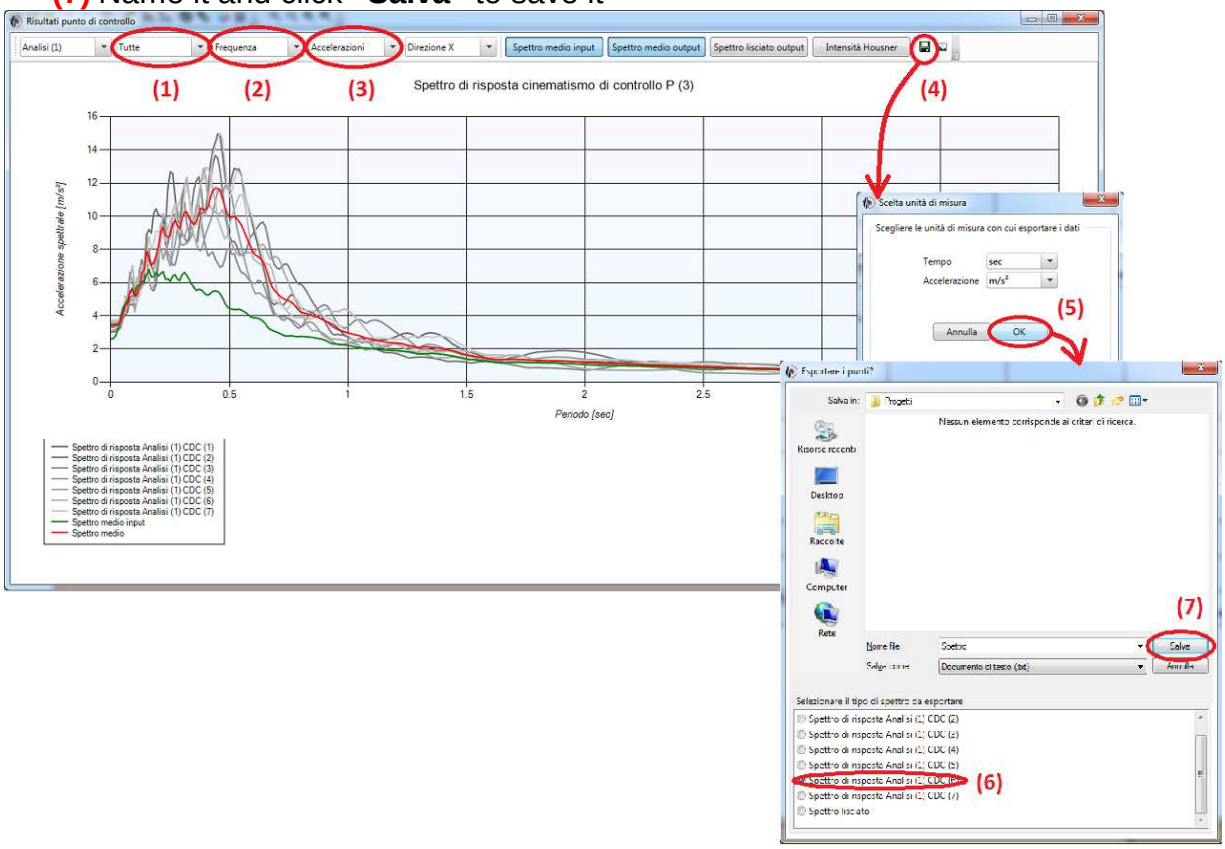


Fig. 4.4 – NTC2008 elastic response spectrum.

5.3 Viewing column results

You can get results in terms of displacements, velocities, accelerations, deformations and stresse for a check column.
For example to get X accelerations of check column in fig. 3.5 select “Accelerazione X”.

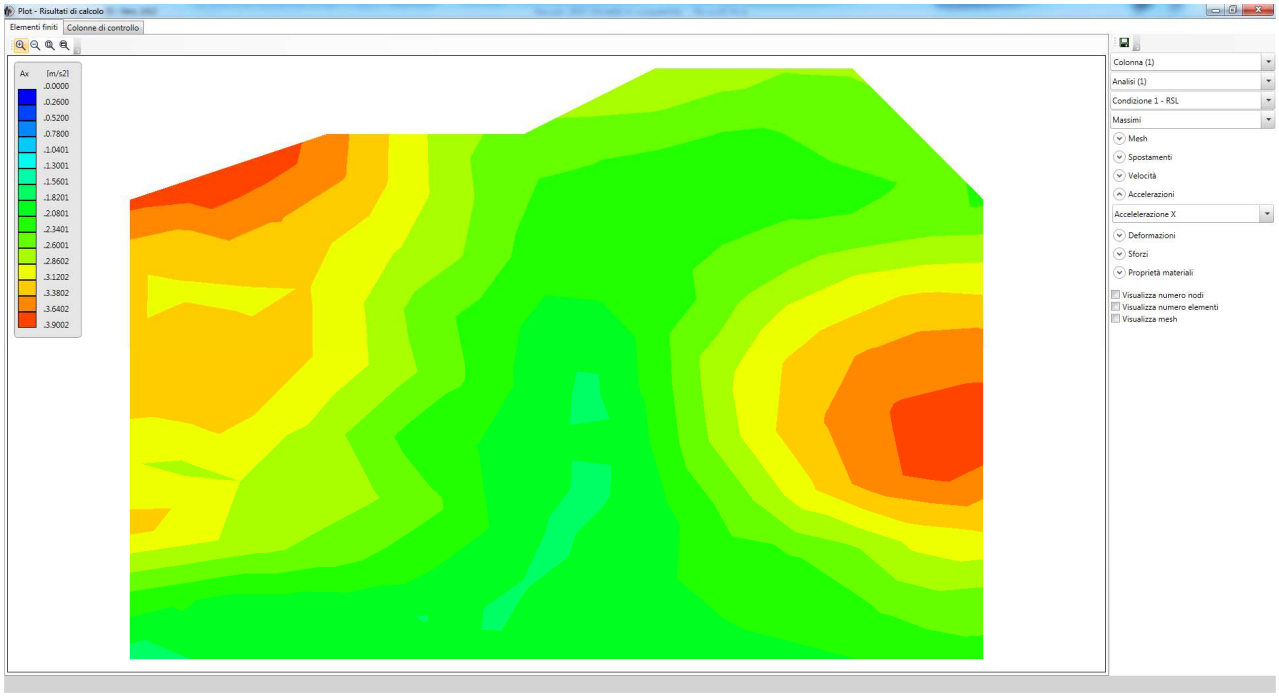


Fig. 3.5 – Viewing check column results.

Select tab-page “**Colonna di controllo”** of plot window to view the graph (fig.3.6). Graphs can be saved.

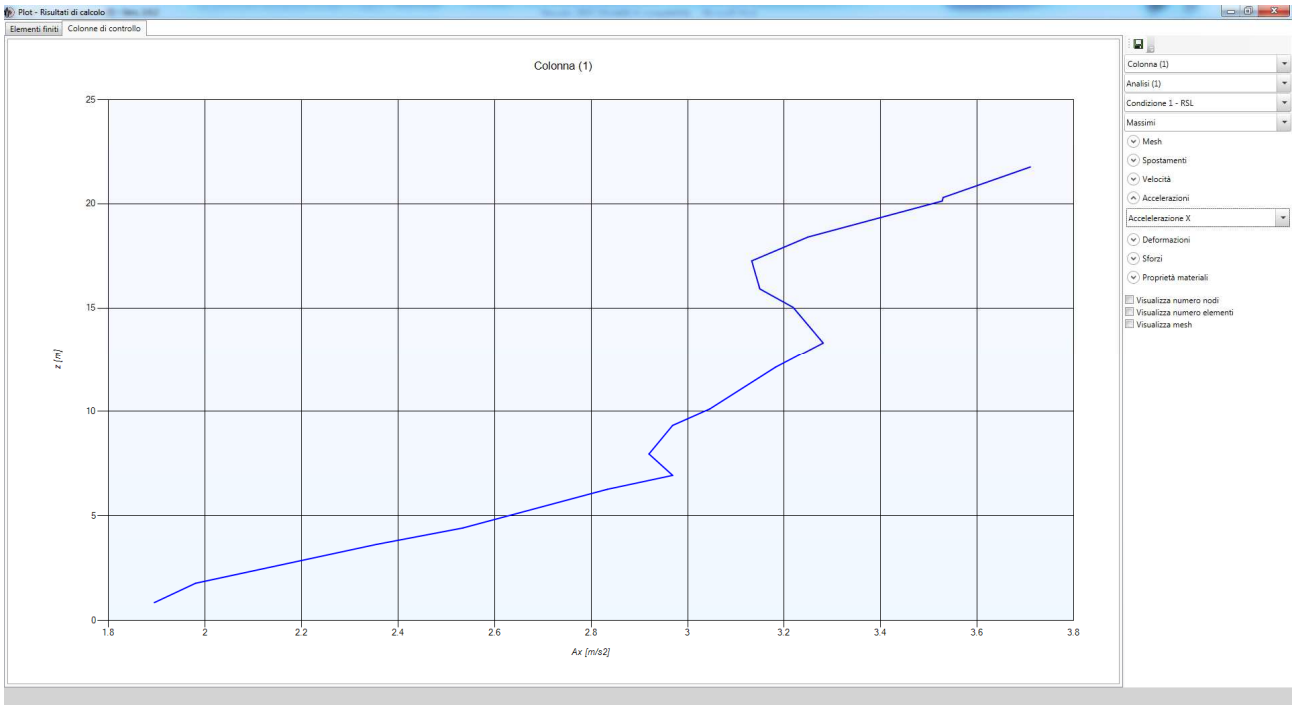


Fig. 3.6 – Check column X displacement plot.

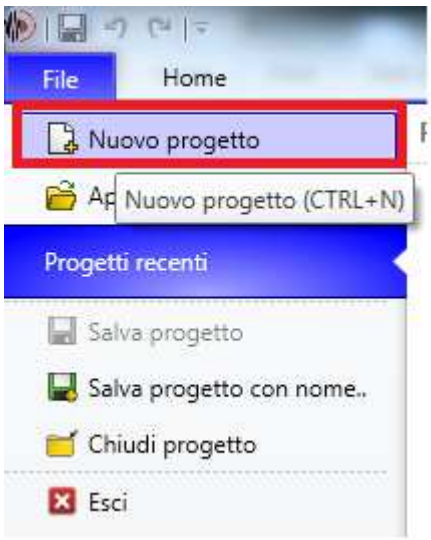
5.4 Calculation report

Once you complete the processing you can generate calculation report by clicking 

in the main window.

6 Example 1. Input a model for a 1D analysis

Step 1. Create new project
From menu file select “Nuovo progetto”



Select “Salva progetto con nome” to save as.

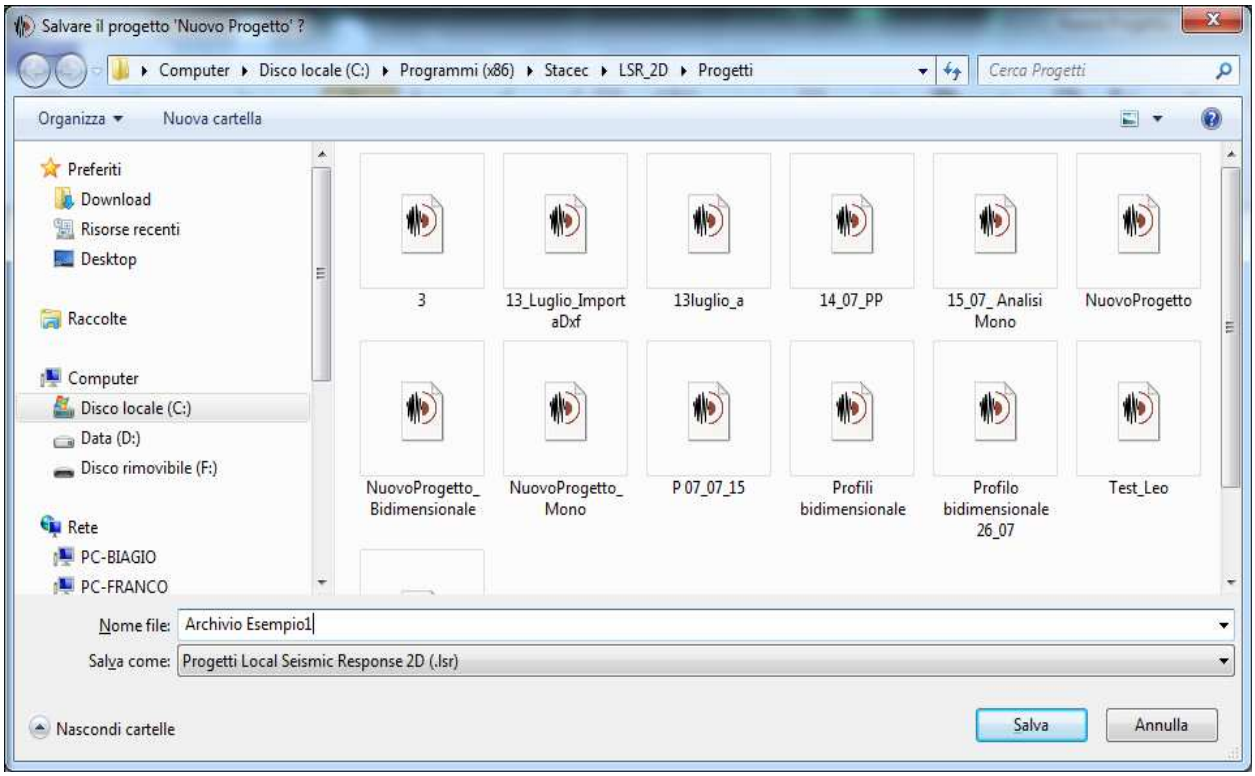
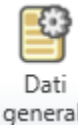


Fig. 6.1 – Saving project.

Step 2. General data defining



Open general data by clicking .
Set values as showed in fig. 6.2 and fig. 6.3 and click “ok” to confirm.

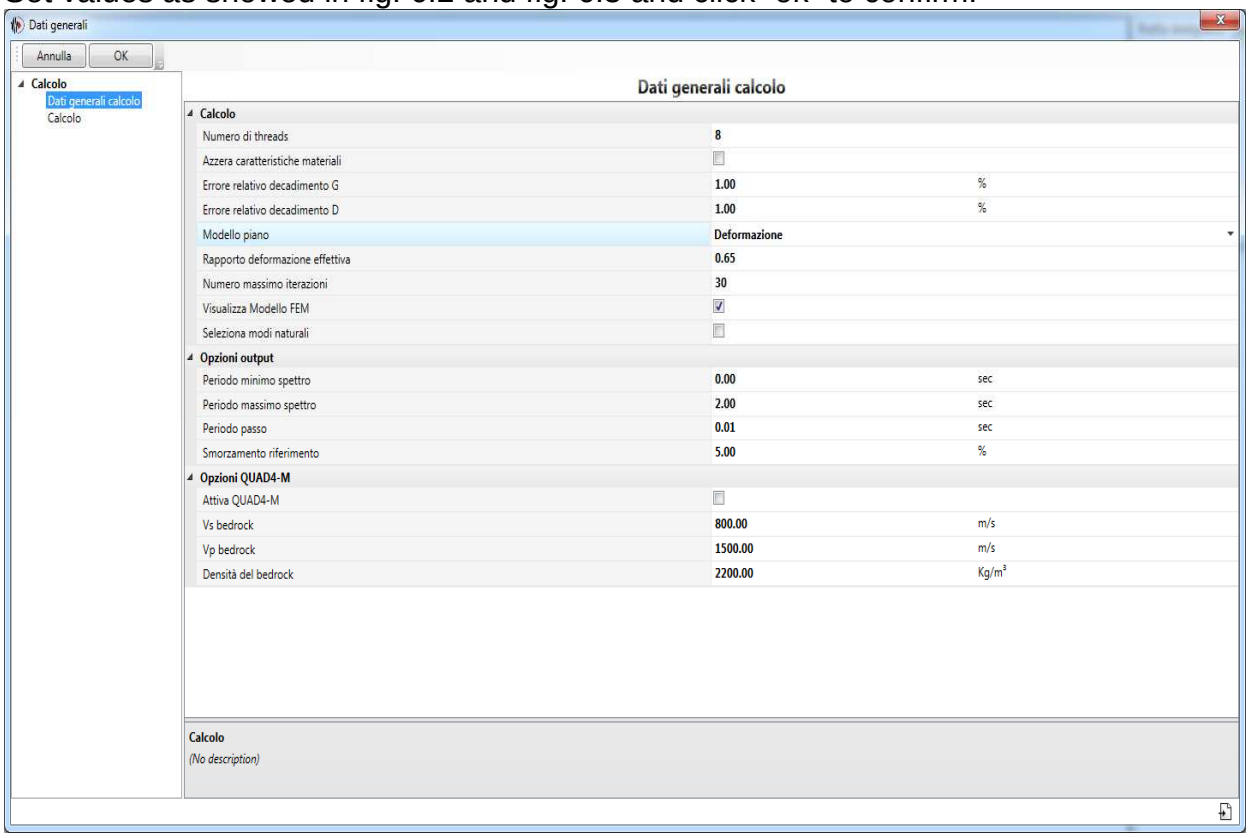


Fig. 6.2 – General data setting.

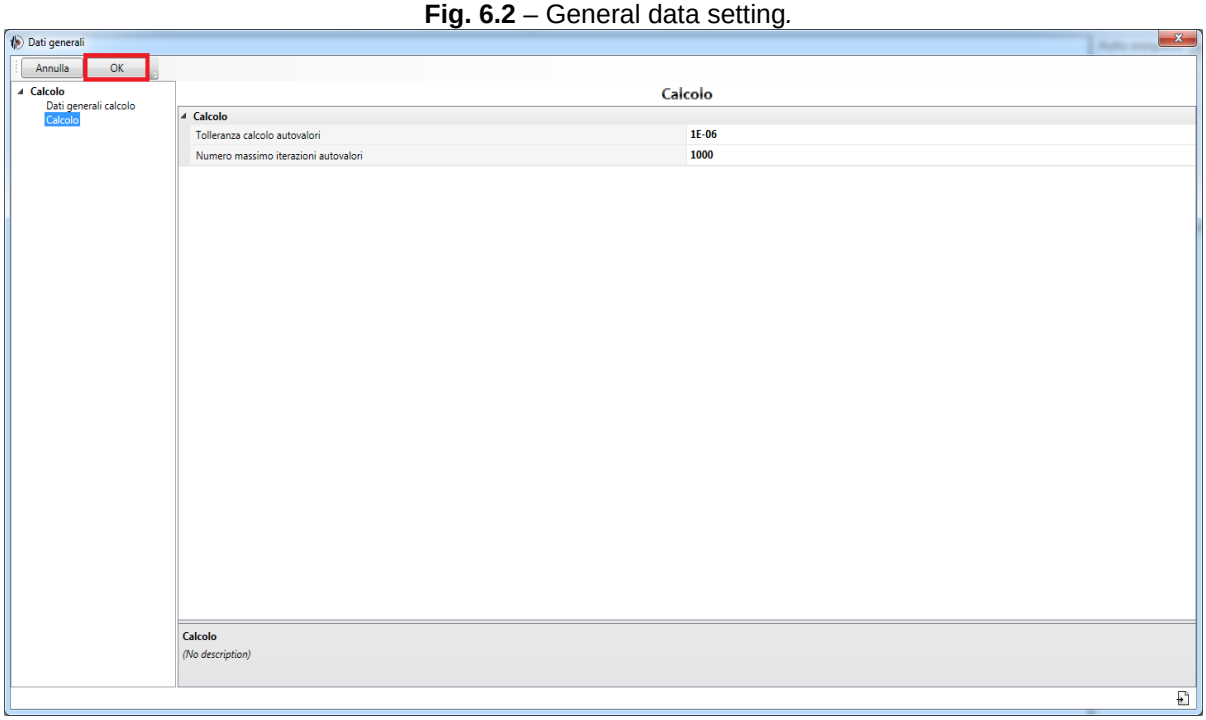


Fig. 6.3 – General data setting.

Number of threads should be set to the number of CPUs of your hardware.

Step 3. Design acceleration time curves.

Let's consider 2 analyses.

1. "SLV" (life safety limit state)
2. "SLD" (damage limit state)

Let Ag_SLV= 0.264g e Ag_SLD= 0.091g in accordance to Italian code NTC 2008.

Click  set PGA = 2.59 m/s^2 for "SLV" analysis. Confirm by clicking "OK".

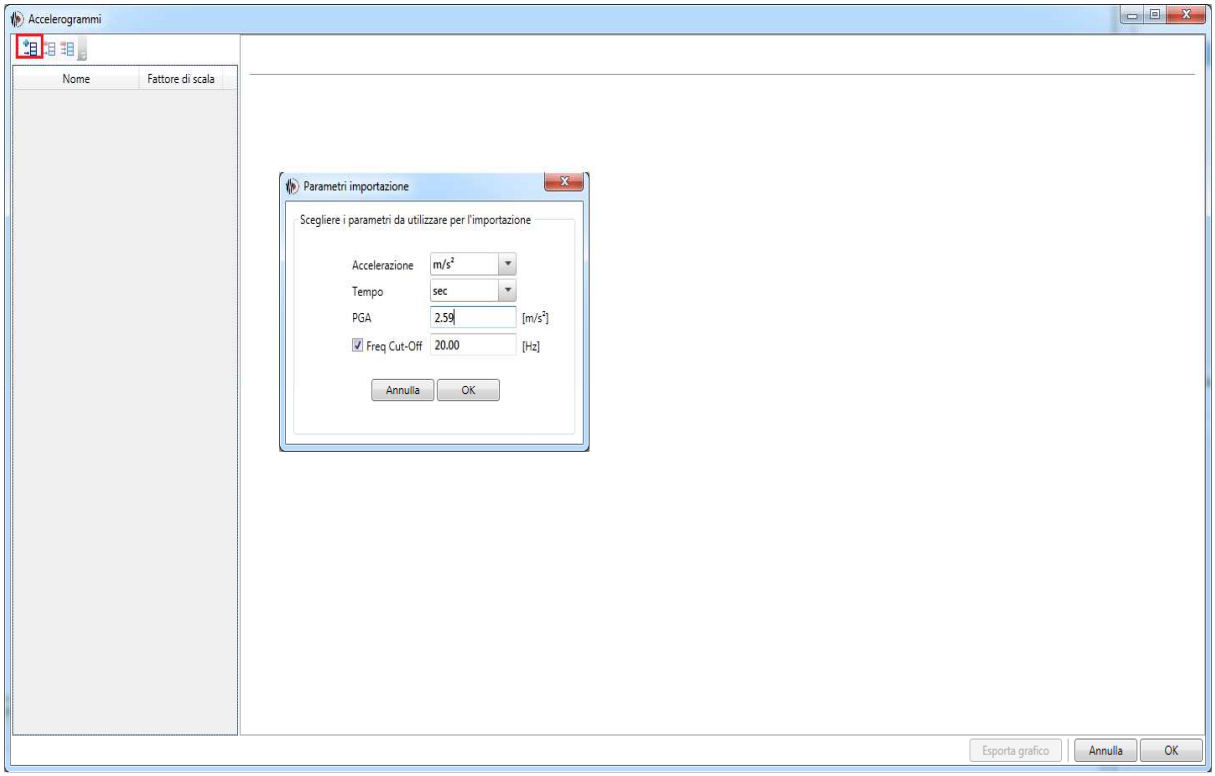


Fig. 6.4 – Defining acceleration time curve.

Choose a compatible set of 7 acceleration-time curves for "SLV" analysis using multiple selection and confirm by clicking "Apri" (fig. 5.5) .

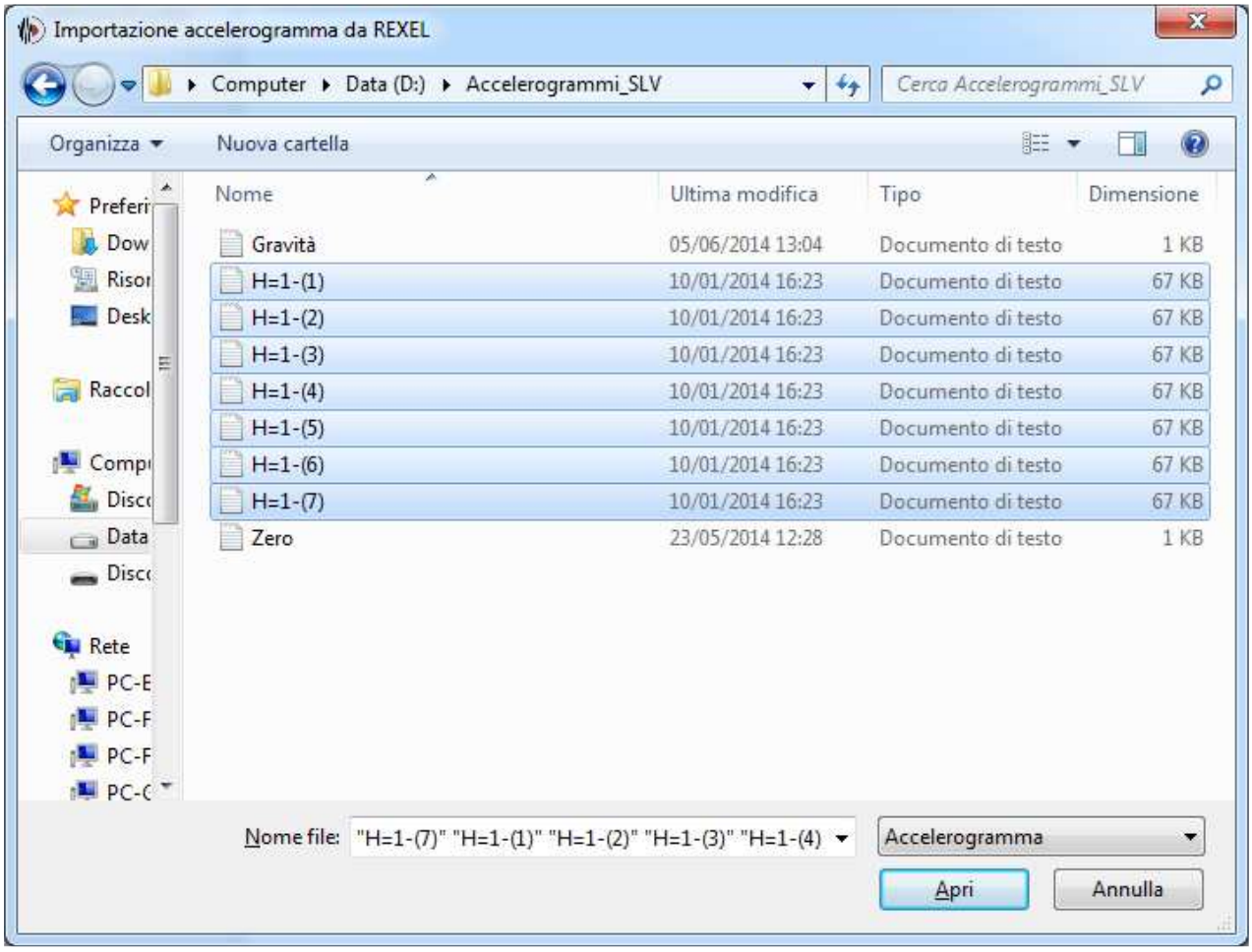
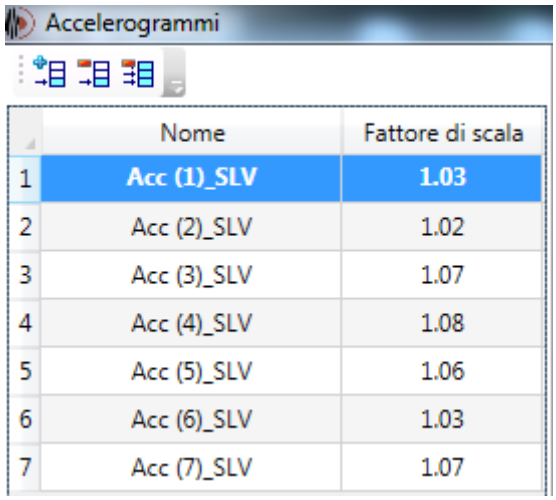


Fig. 6.5 – Defining acceleration time curve.

Rename curves as in fig 5.6



	Nome	Fattore di scala
1	**Acc (1)_SLV**	**1.03**
2	Acc (2)_SLV	1.02
3	Acc (3)_SLV	1.07
4	Acc (4)_SLV	1.08
5	Acc (5)_SLV	1.06
6	Acc (6)_SLV	1.03
7	Acc (7)_SLV	1.07

Fig. 6.6 – Defining acceleration time curve

.

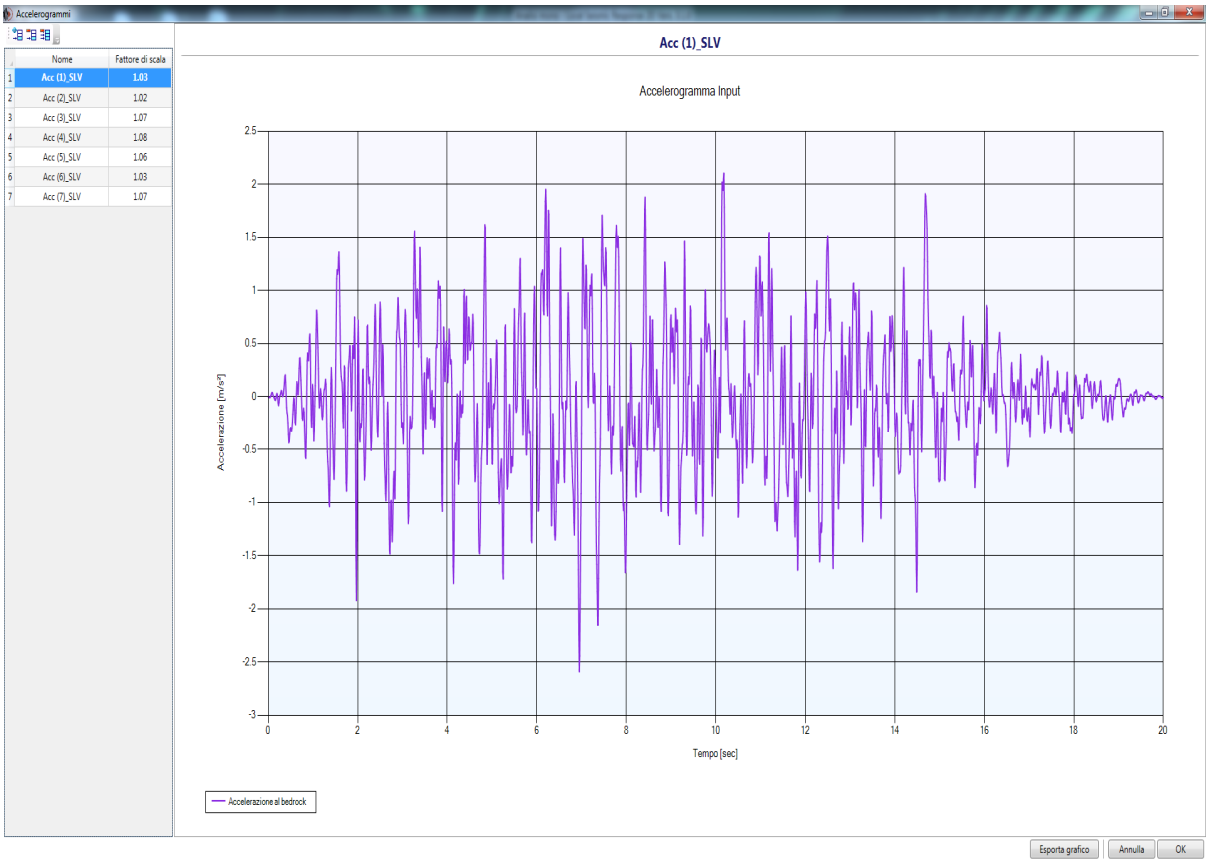


Fig. 6.7 – Defining acceleration time curve

Repeat the previous operations for “SLD” analysis with PGA = 0.256 m/s^2.

Fig. 6.8 – acceleration time curve parameters

.

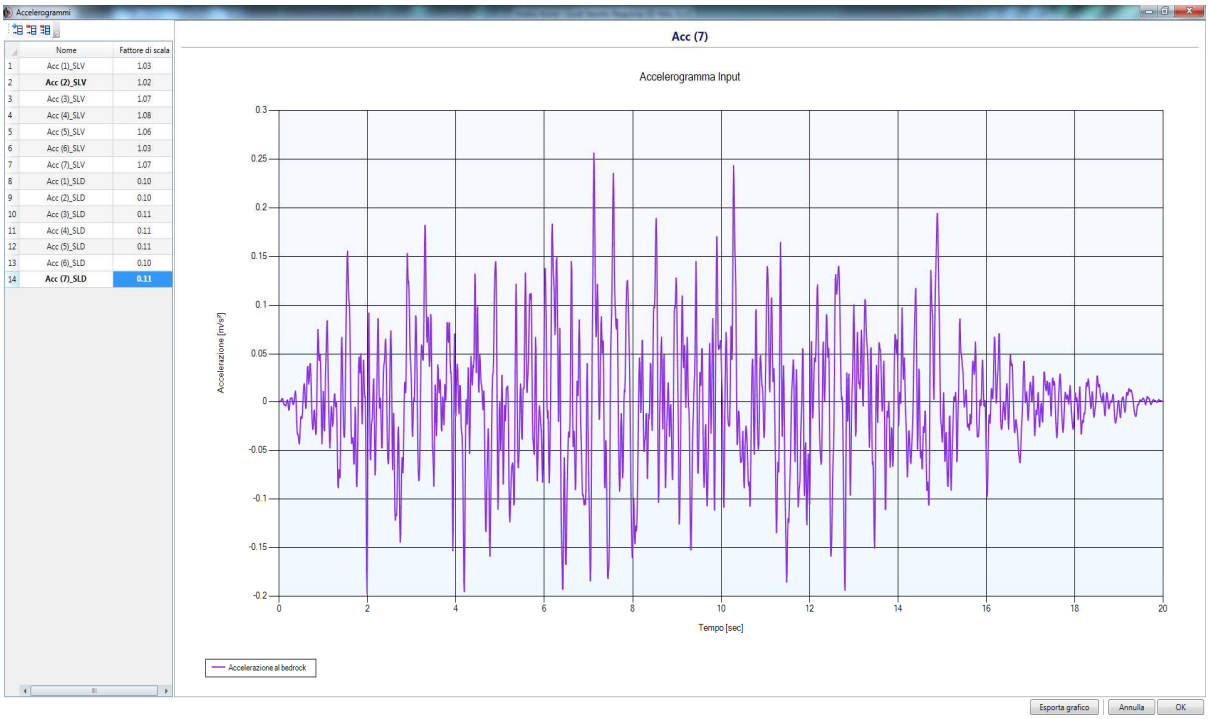
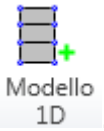


Fig. 6.9 – Defining acceleration time curve

Confirm by clicking “OK”.

Step 4. Stratigraphy definition

Click 

. Set values as showed in fig. 5.10 and click “ok” to confirm.

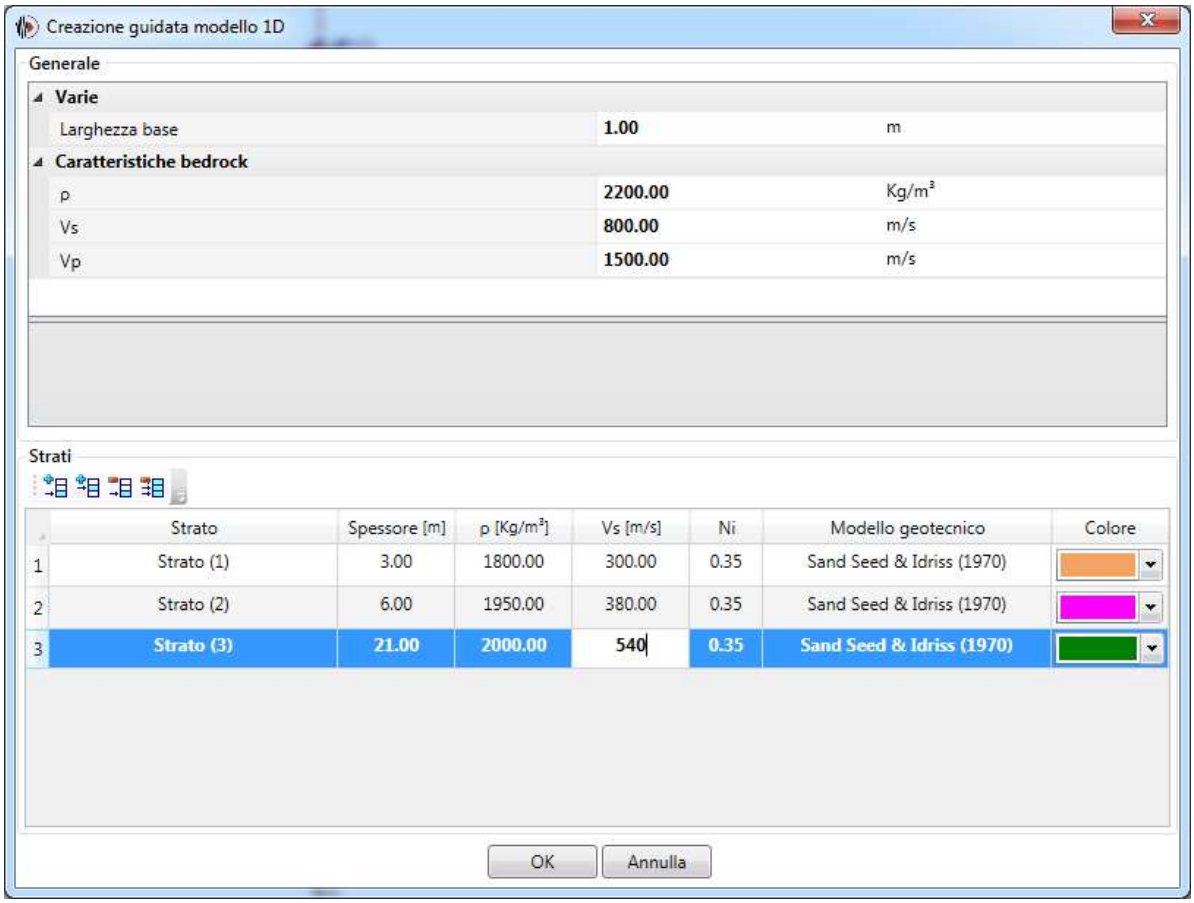


Fig. 6.10 –1D Modeling.

Step 5. Analyses defining

Add 2 analyses.

1. "Analisi (1)_SLV" (life safety limit state)
2. "Analisi (1)_SLD" (damage limit state)

For each analysis define 7 load cases.

Fig. 6.11 – Load case definition.

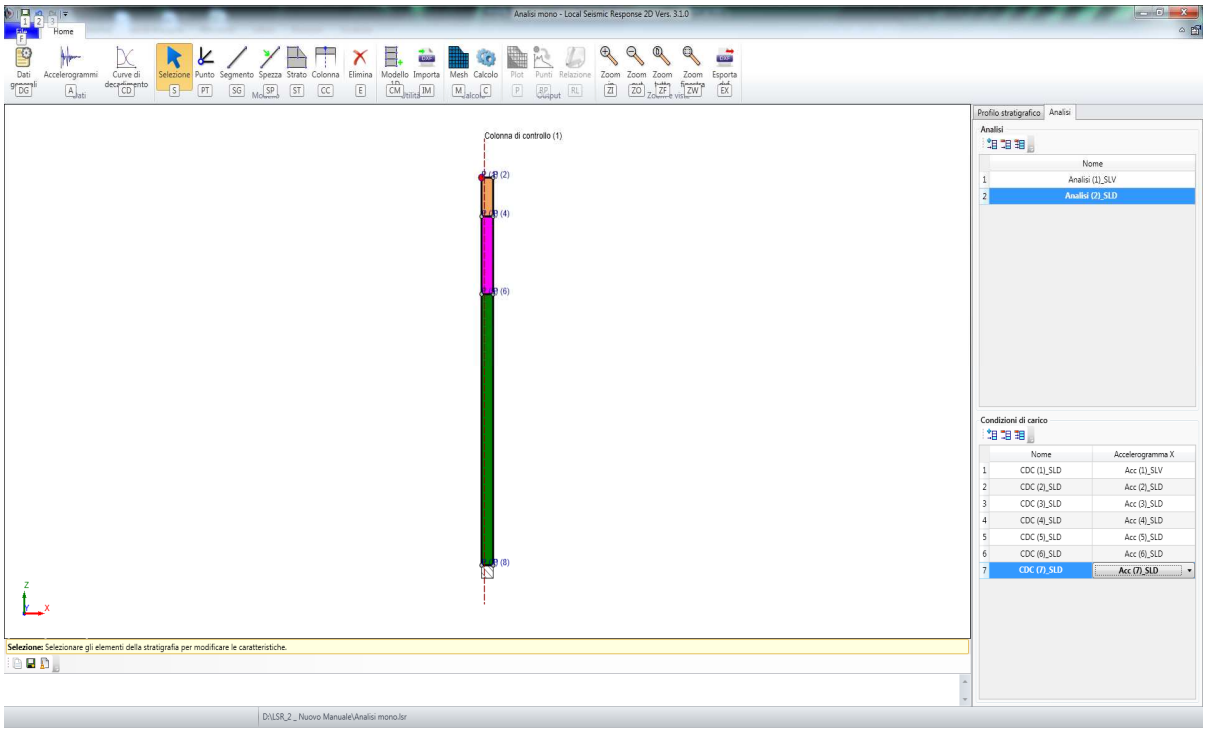


Fig. 6.12 – Input analisi.

Step 6. Check column and check point

In 1D modeling the software define automatically them.

7. Example 2. Input a model for a 2D analysis

Step 1 and 2 as in the previous example.

Step 3. Design acceleration time curves.
Let's consider 2 analyses.

1. "SLV" (life safety limit state)
2. "SLD" (damage limit state)

Let Ag_SLV= 0.264g e Ag_SLD= 0.091g in accordance to Italian code NTC 2008.

Click , set PGA = 2.59 m/s^2 for "SLV" analysis. Confirm by clicking "OK".

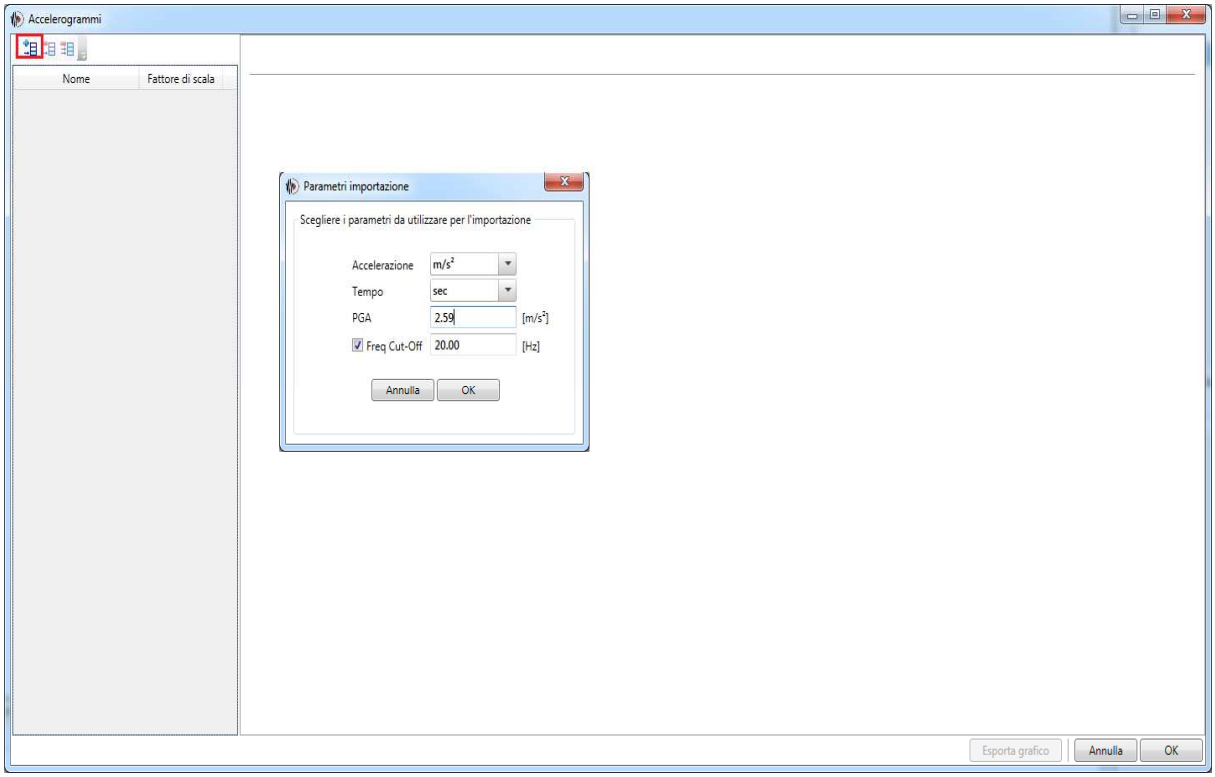


Fig. 7.1 – Defining acceleration time curve

Choose a compatible set of 7 acceleration-time curves for "SLV" analysis using multiple selection and confirm by clicking "Apri" (fig. 7.2) .

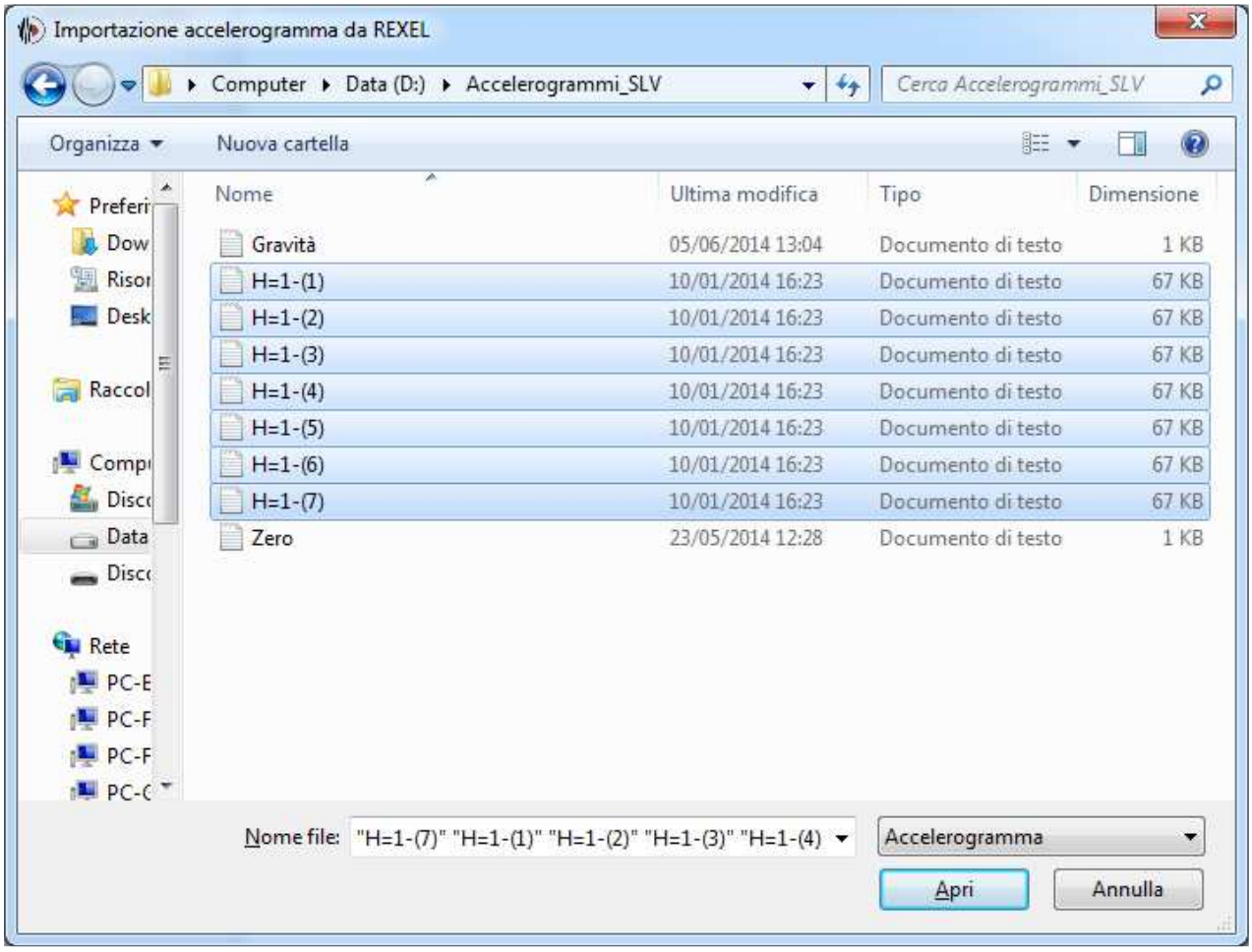
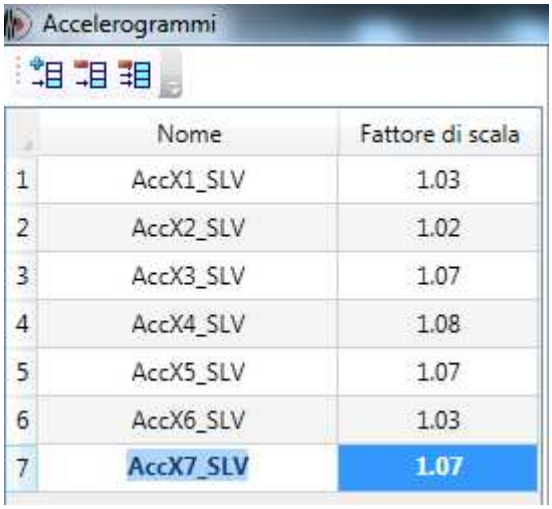


Fig. 7.2 - Defining acceleration time curve



	Nome	Fattore di scala
1	AccX1_SLV	1.03
2	AccX2_SLV	1.02
3	AccX3_SLV	1.07
4	AccX4_SLV	1.08
5	AccX5_SLV	1.07
6	AccX6_SLV	1.03
7	AccX7_SLV	1.07

Fig. 7.3 - Defining acceleration time curve.

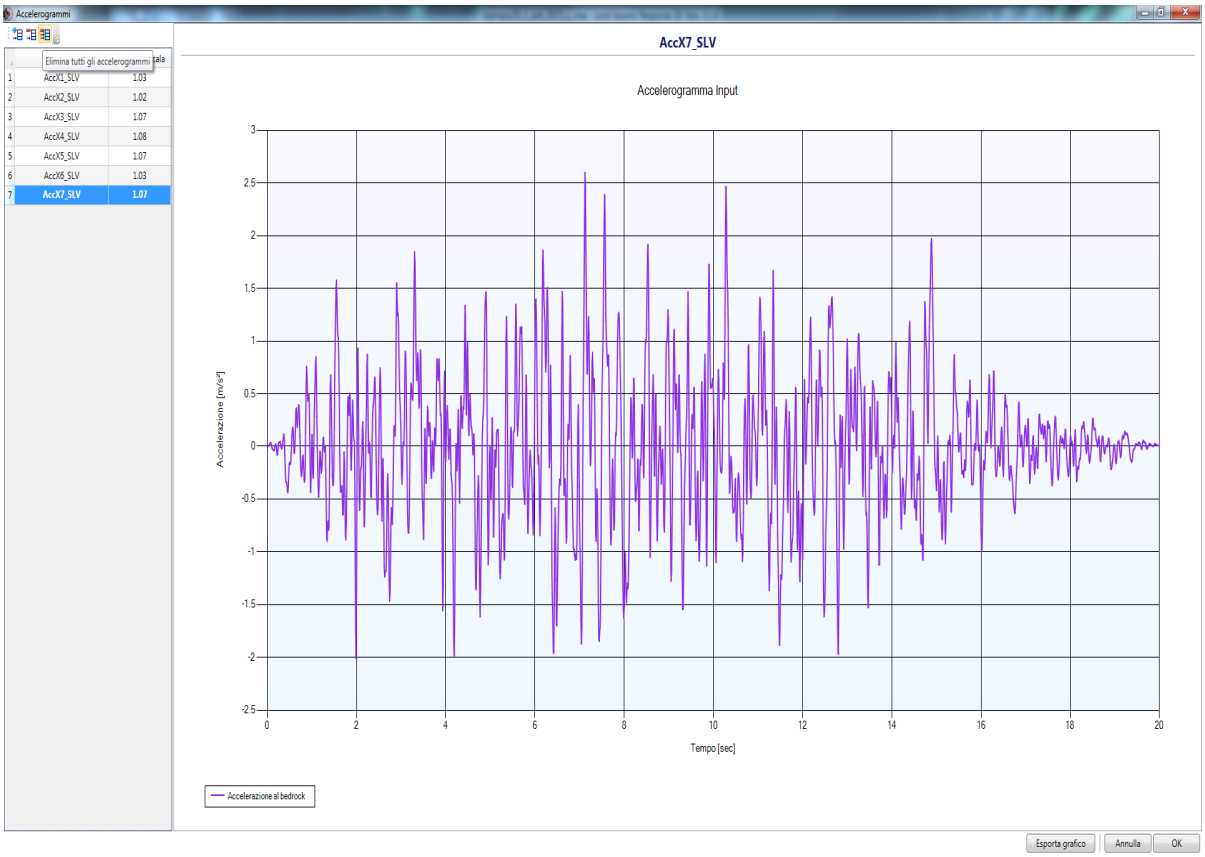


Fig. 7.4 - Defining acceleration time curve

Repeat the previous operations for “SLD” analysis with PGA = 0.256 m/s^2.

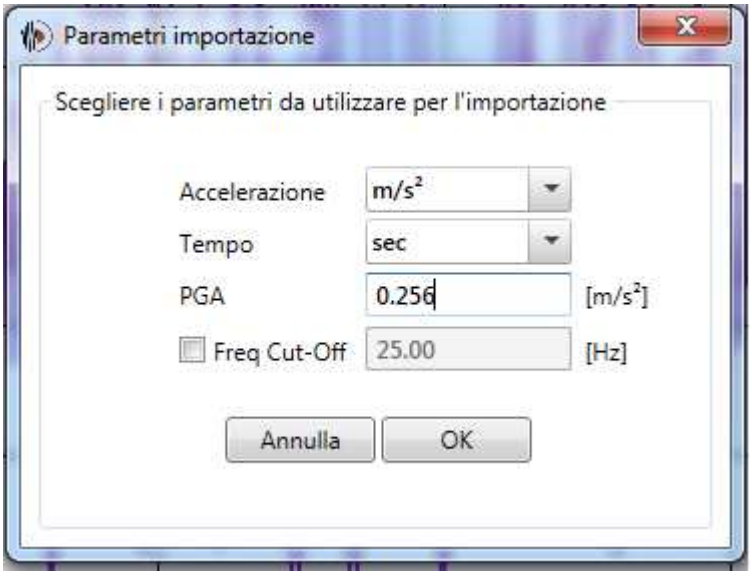


Fig. 7.5 – Defining acceleration time curve

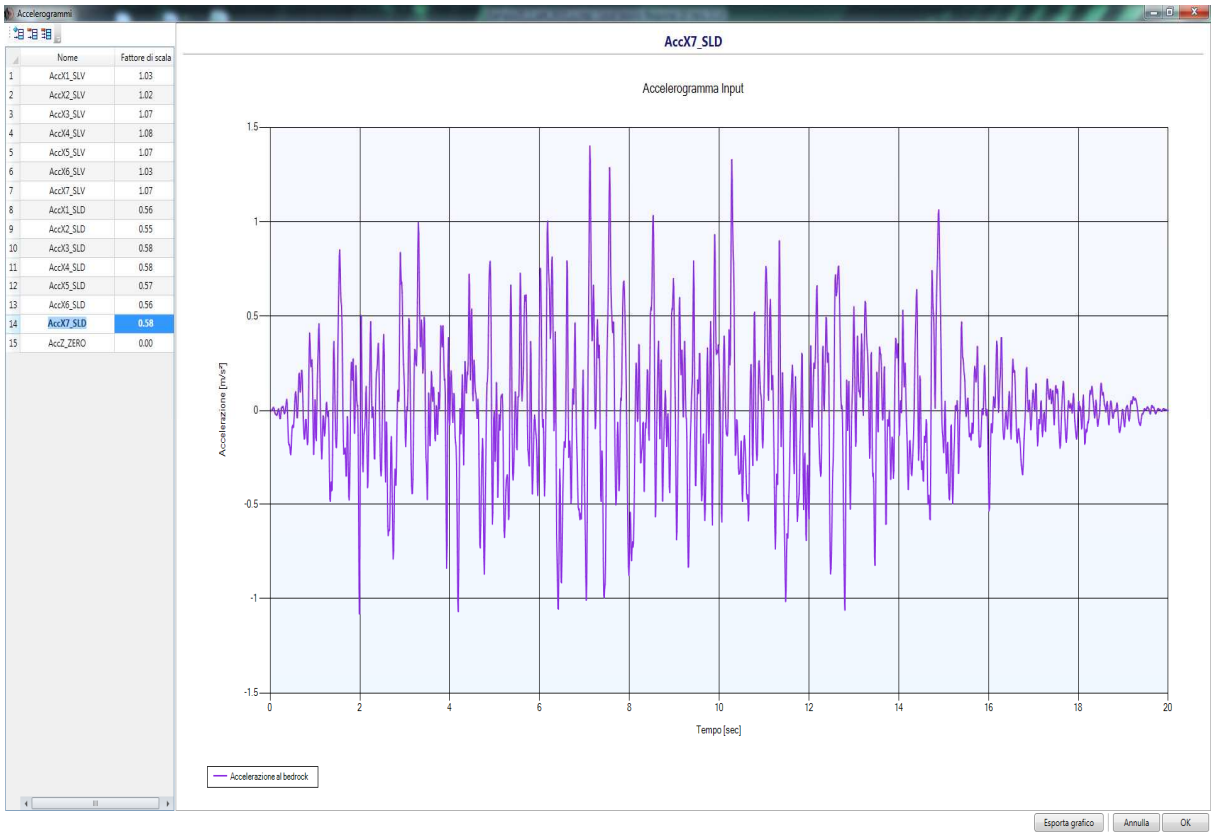
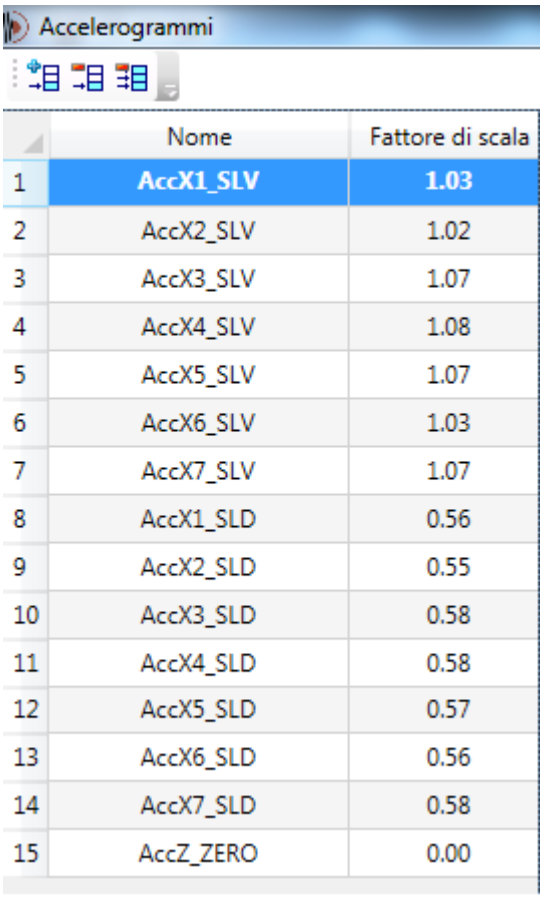


Fig. 7.6 - Defining acceleration time curve



	Nome	Fattore di scala
1	**AccX1_SLV**	**1.03**
2	AccX2_SLV	1.02
3	AccX3_SLV	1.07
4	AccX4_SLV	1.08
5	AccX5_SLV	1.07
6	AccX6_SLV	1.03
7	AccX7_SLV	1.07
8	AccX1_SLD	0.56
9	AccX2_SLD	0.55
10	AccX3_SLD	0.58
11	AccX4_SLD	0.58
12	AccX5_SLD	0.57
13	AccX6_SLD	0.56
14	AccX7_SLD	0.58
15	AccZ_ZERO	0.00

Fig. 7.7 - Defining acceleration time curve

Add a curve “AccZ_ZERO” with a zero scale factor.

Confirm by clicking “OK”.

Step 4. Analyses defining

Add 2 analyses.

1. “SLV” (life safety limit state)
2. “SLD” (damage limit state)

For each analysis define 7 load cases and assign acceleration-time curves previously defined.

Set Z acceleration time curve to “ AccZ_ZERO” so you can consider only horizontal seismic motion.

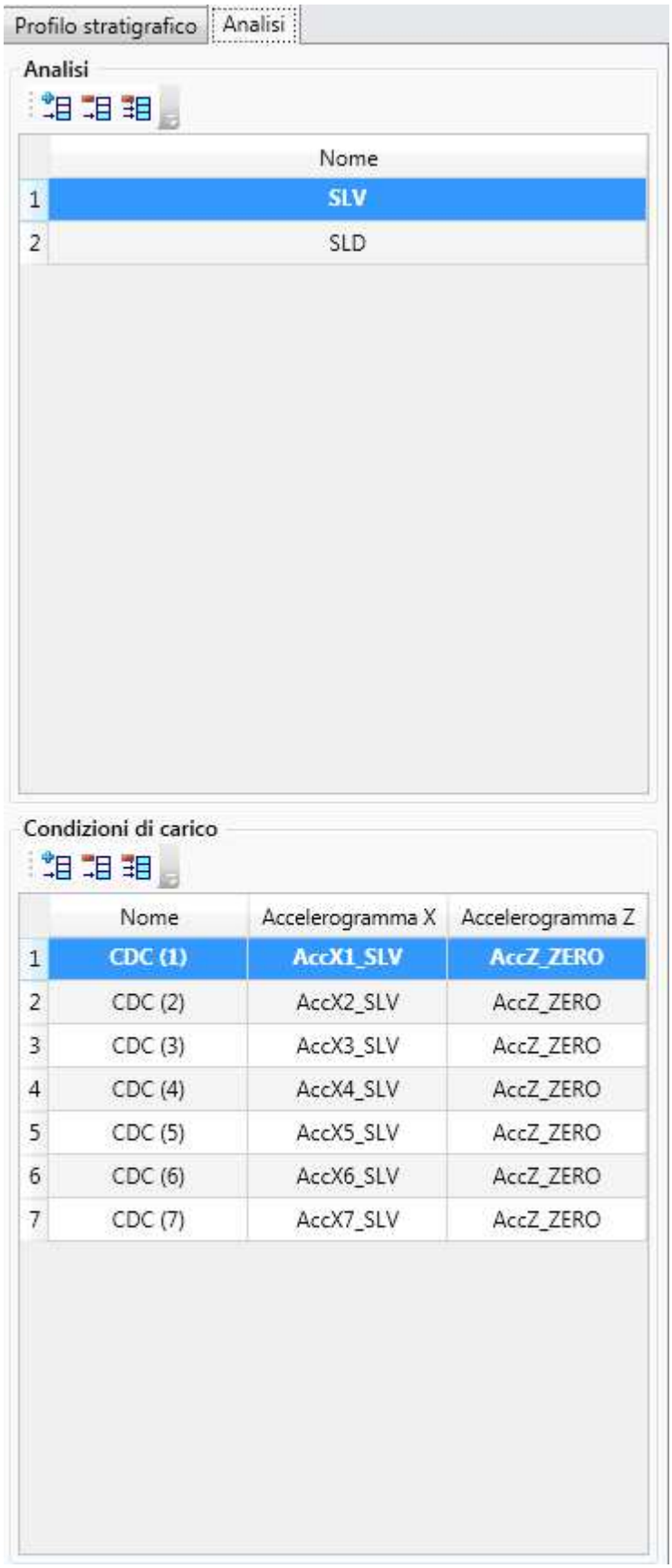


Fig. 7.8 - Defining analyses

Step 5. Stratigraphy definition

Click Punto and add the points:

1. 1 (-10,0);
2. 2 (50,0);
3. 3 (50,30);
4. 4 (30,30);
5. 5 (25,35);
6. 6 (15,30);
7. 7 (10,30);
8. 8 (-10,30);
9. 9 (-10,15);
10. 10 (50,18);
11. 11 (24,28);
12. 12 (20,35).

Click Segmento and add the segments:

1. 1-2,
2. 2-10,
3. 10-9,
4. 9-1,
5. 9-8,
6. 8-7,
7. 7-6,
8. 6-12,
9. 12-5,
10. 5-4,
11. 4-3,
12. 3-10,
13. 4-11,
14. 11-7.

Click Strato, click in the inside of the polygon 7-6-12-5-4-11-7. First layer will be added. Assign to it parameters as showed in fig. 7.10

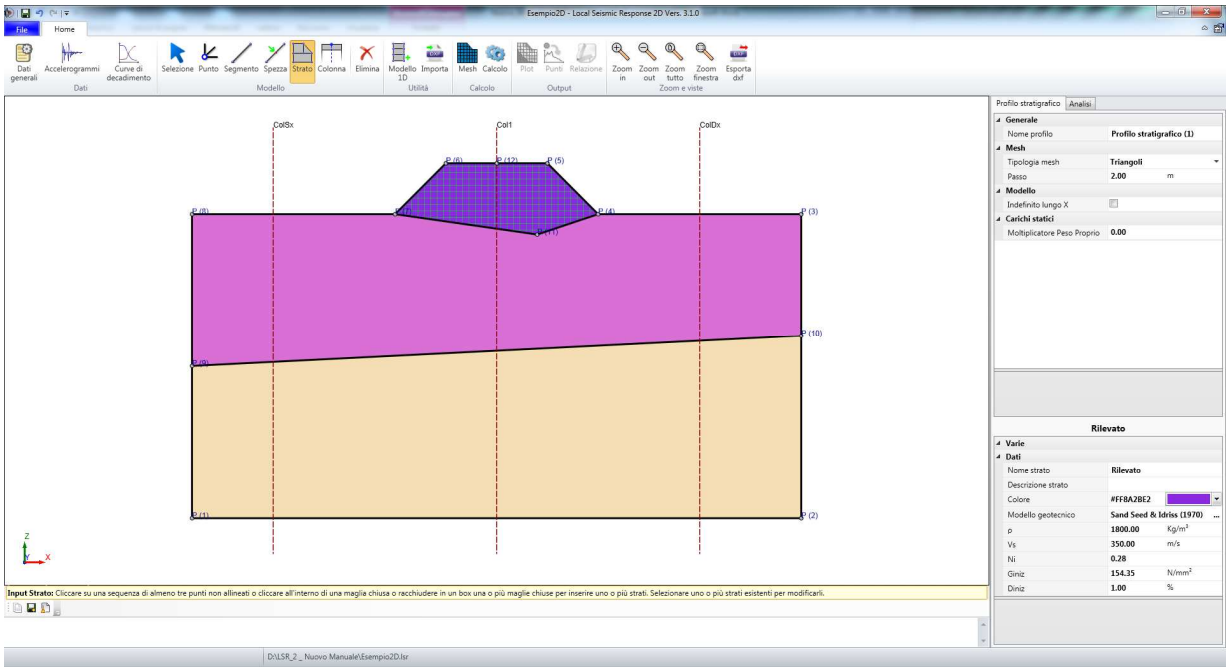


Fig. 7.9 Defining layers

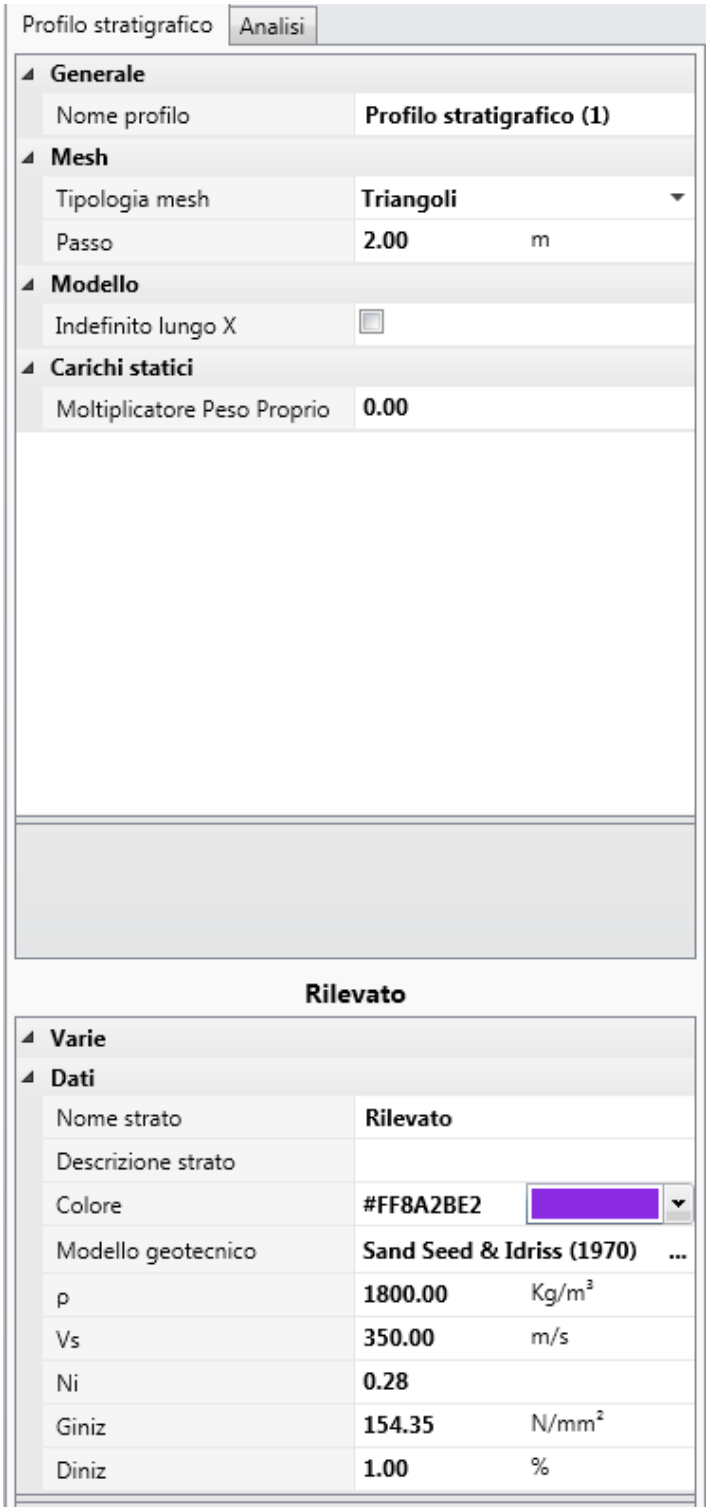


Fig. 7.10 – Defining layer properties

Click in the inside of the polygon 8-7-11-4-3-10-9-8. Assign to it parameters as showed in 7.11.

Fig. 7.11 - Defining layer properties

Click in the inside of the polygon 10-2-1-9. Assign to it parameters as showed in fig. 7.12

Fig. 7.12 – Defining layer properties

Line segment 1-2 is the bedrock boundary. Select it, check “bedrock” and assign properties as in fig. 7.13.

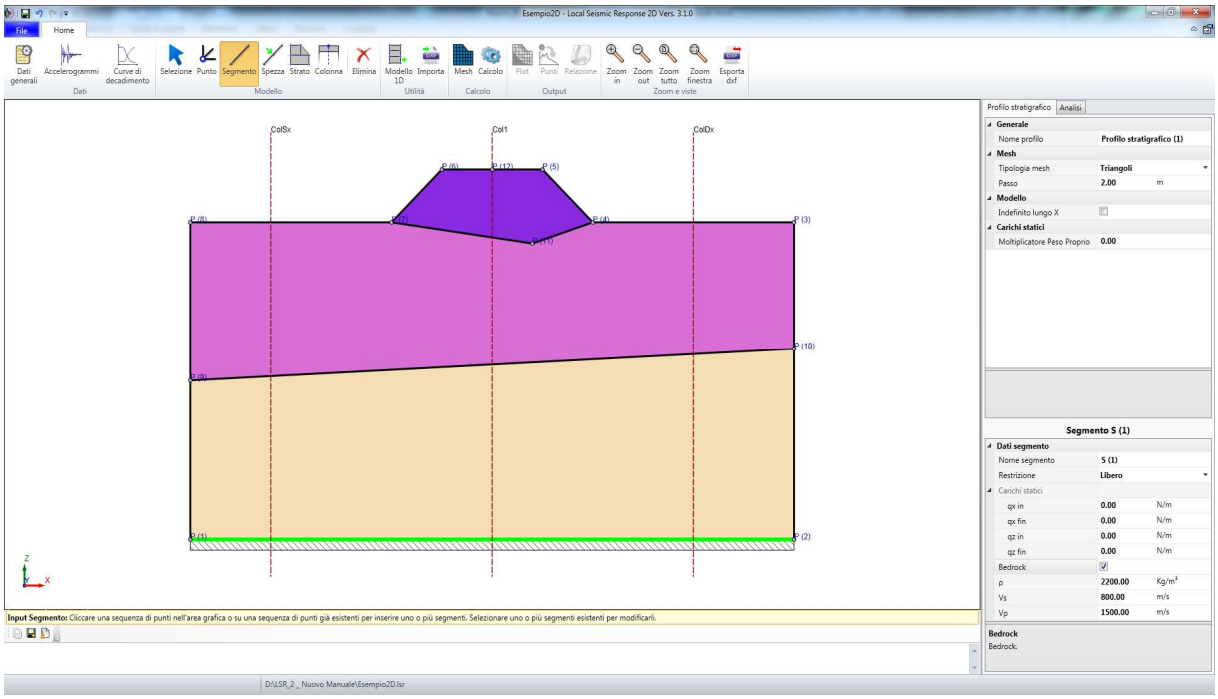


Fig. 7.13 - Restraints

Select line segments 1-2, 1-9, 9-8, 2-10 and 10-3 (fig. 7.14) and apply Z restraint ("Vincolato Z").

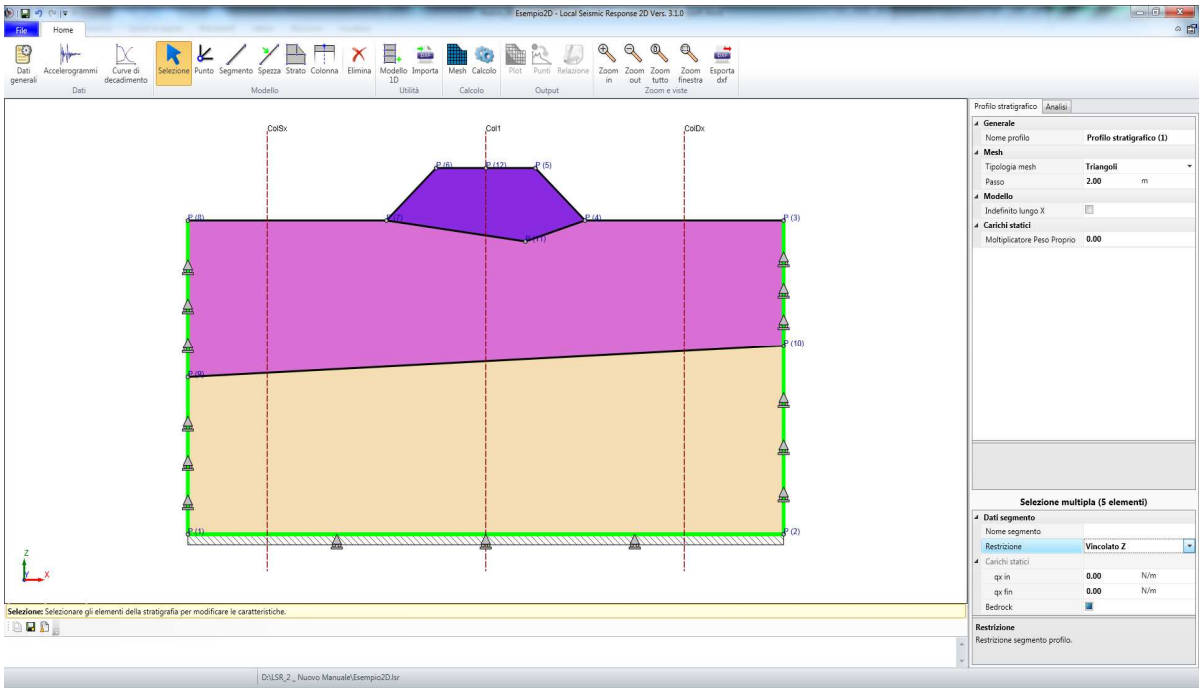


Fig. 7.14 - Restraints

Step 6. Check column

Click Colonna and add 3 check columns, one on the left of the embankement, one in the middle, and one on the right (see fig. 7.15).

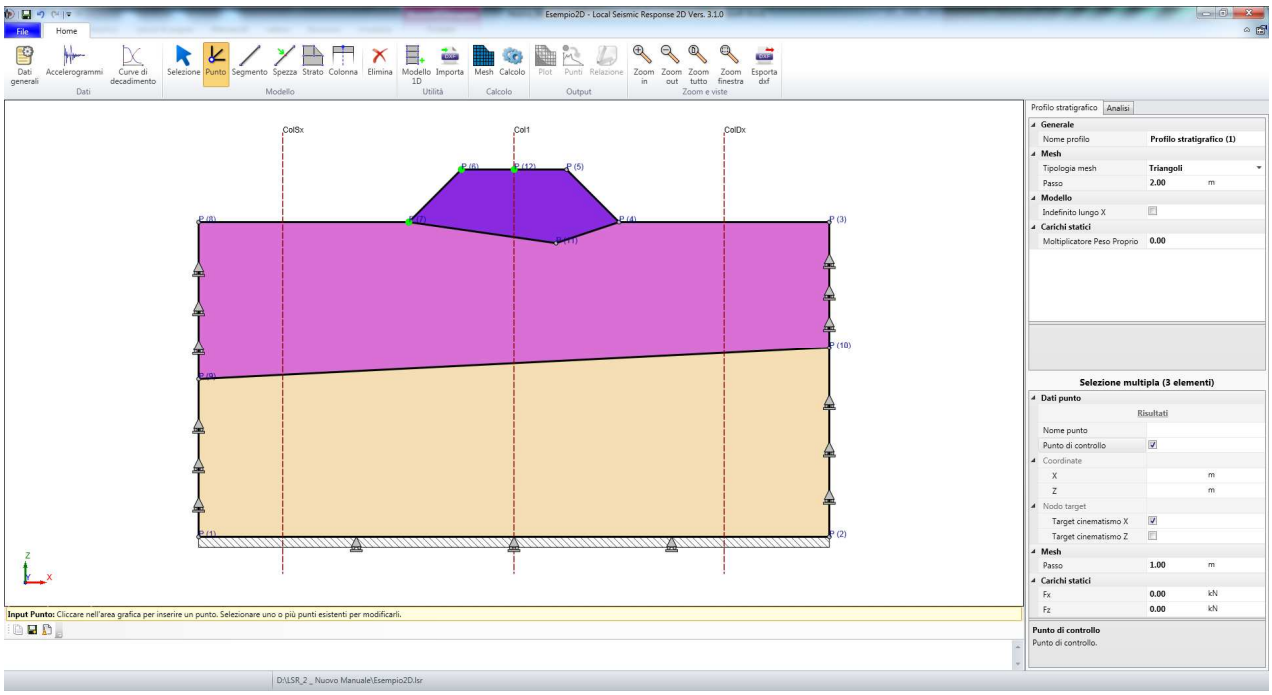


Fig. 7.15 – Check column

Step 7. Check point

Click Punto, select points 6, 7 and 12 and check “Punto di controllo”.

8. Exporting of output

8.1 Exporting the plot of variation of horizontal acceleration with depth along check column

Click to open plot window.

Click tab-page "**Colonna di controllo".** Select column ("Col 1") ,analysis ("SLV"), load case ("Condizione 6 – RSL"), horizontal acceleration ("Accelerazione X") and click  (fig. 8.1)**.** Showed plot will be saved to a .txt file.

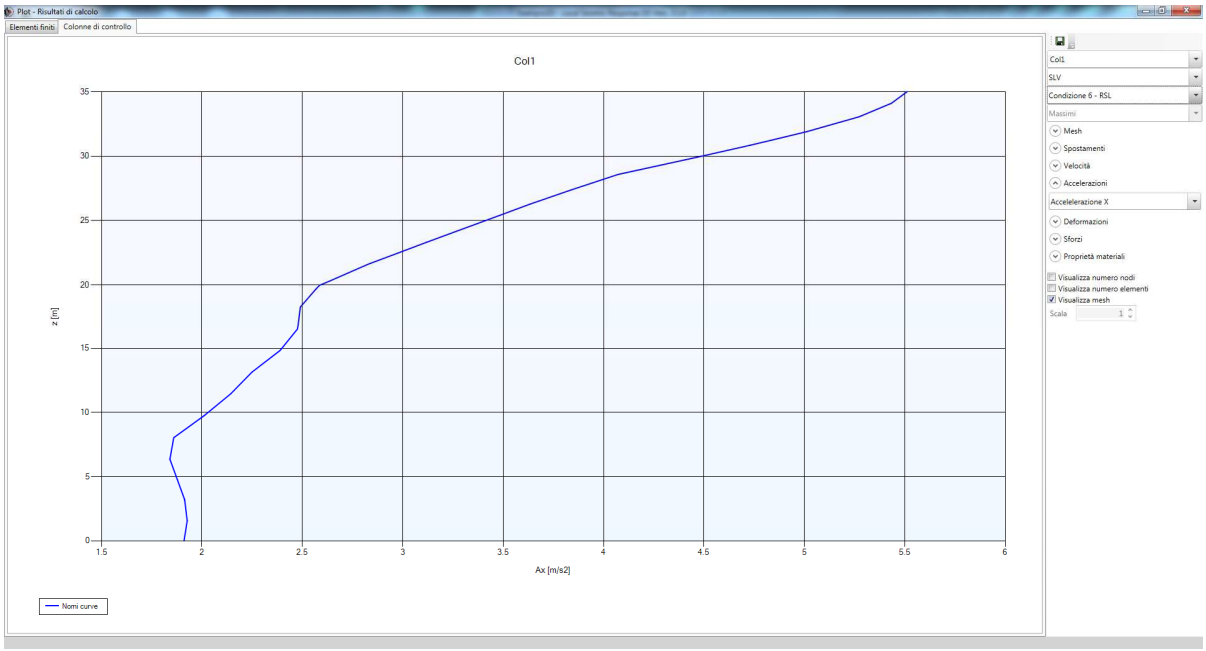


Fig. 8.1 – Saving check column result plot

8.2 Exporting smoothed horizontal acceleration spectrum

Click to open check point results window Select check point ("P6"), analysis ("SLV"), all load cases ("Tutte"), domain ("frequenze"), spectrum type

("accelerazione"), direction ("X"), smoothed output spectrum ("Spettro output lisciato") and smoothing number (10), then click 

(Figg. 8.2 e 8.3).

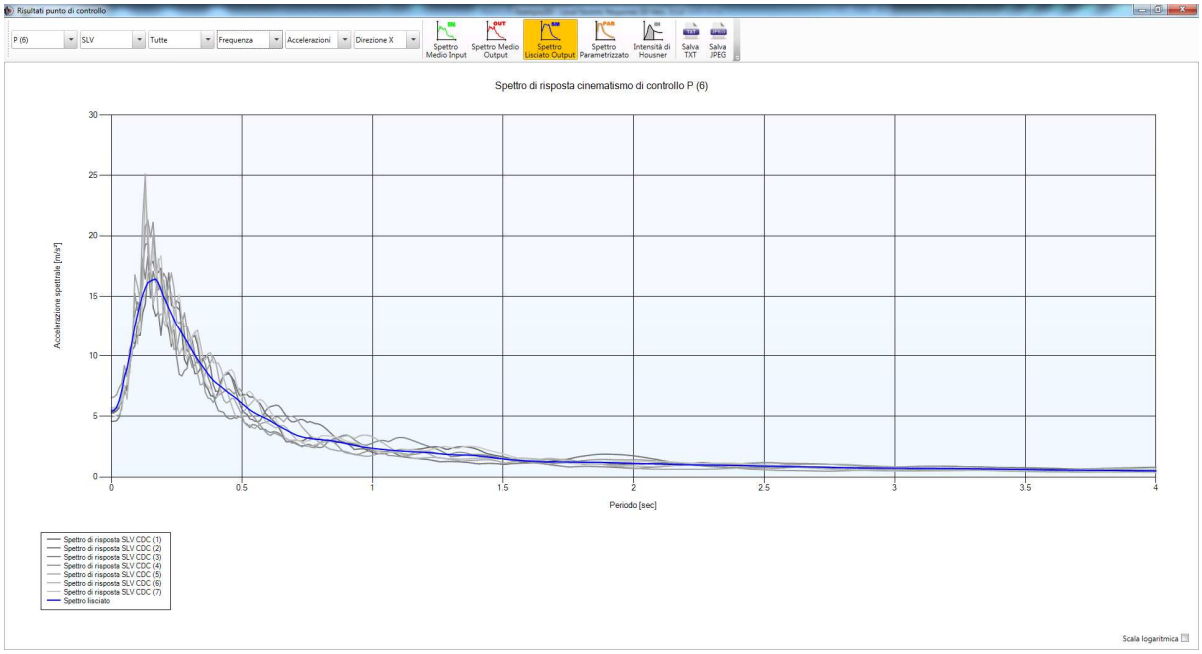


Fig. 8.2 – Check point output

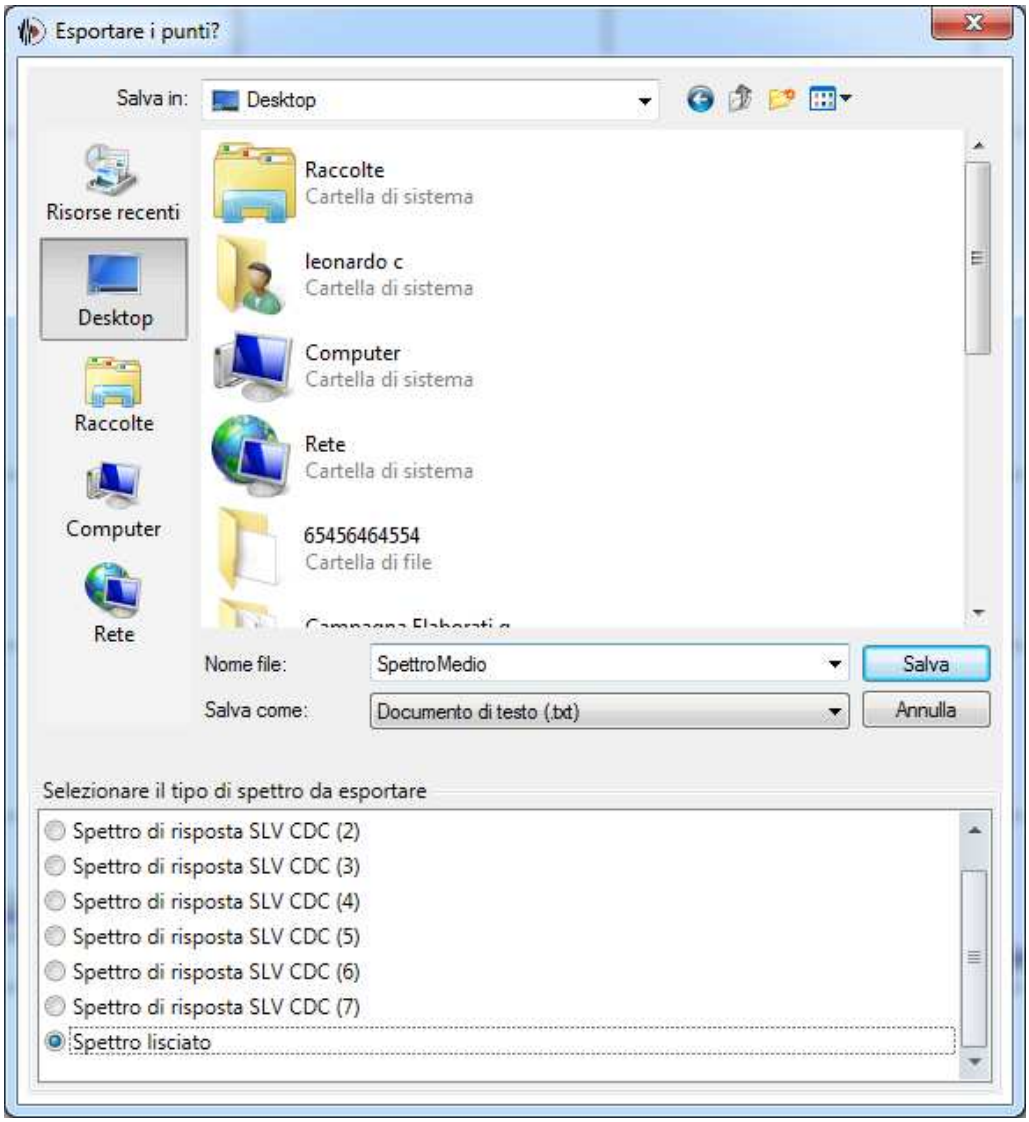


Fig. 8.3 – Saving spectrum points