NextFEM **Designer**

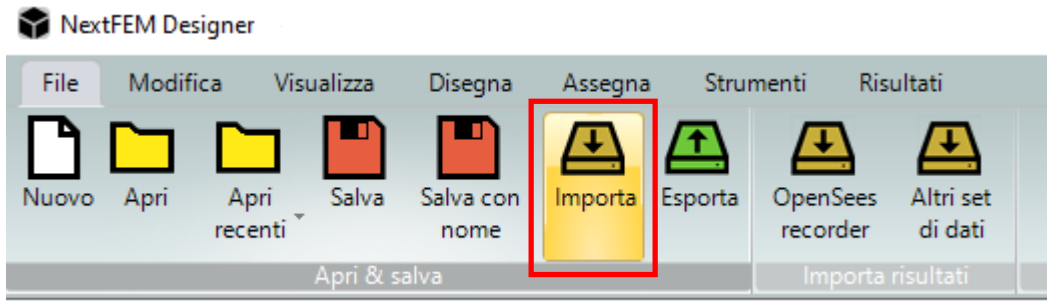
Operating manual

ImportExport Guide

Rev.01 - April 2022

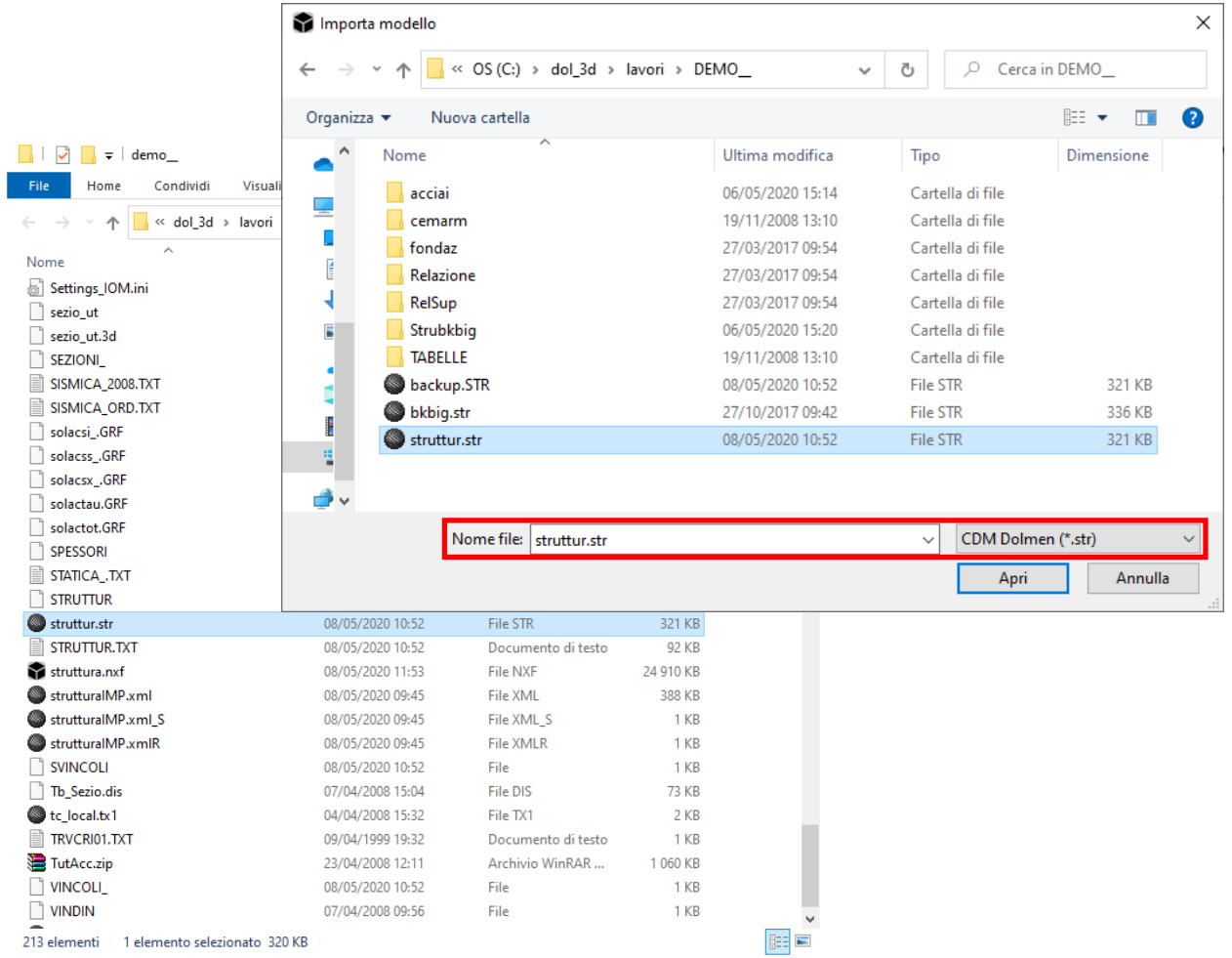
Import from CDM DOLMEN

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu



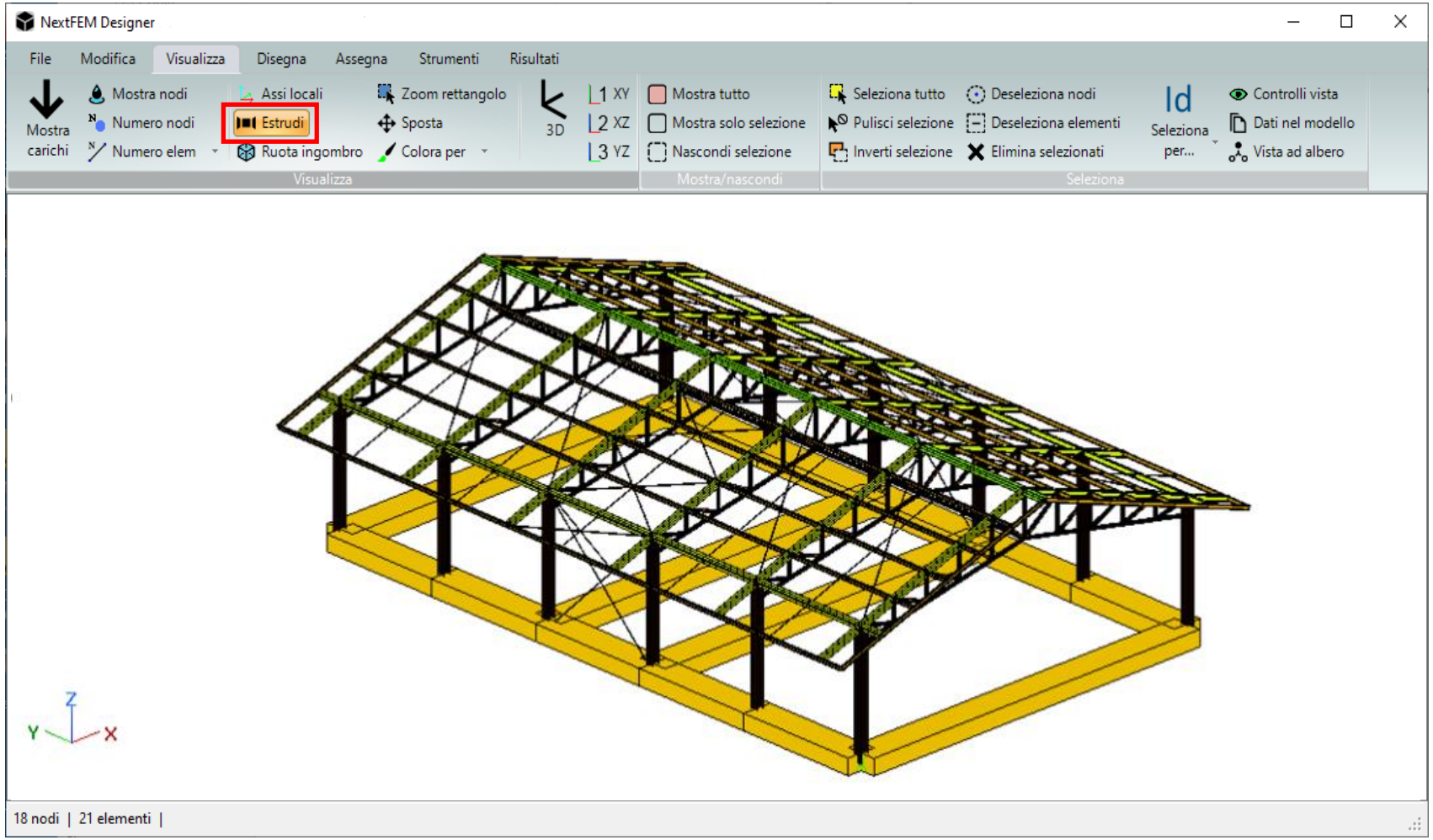
N.B. The model must have already been calculated in
DOLMEN

2. Select *modelname.str (*.str)* as the file type from the drop-down list or drag the file into the viewport.



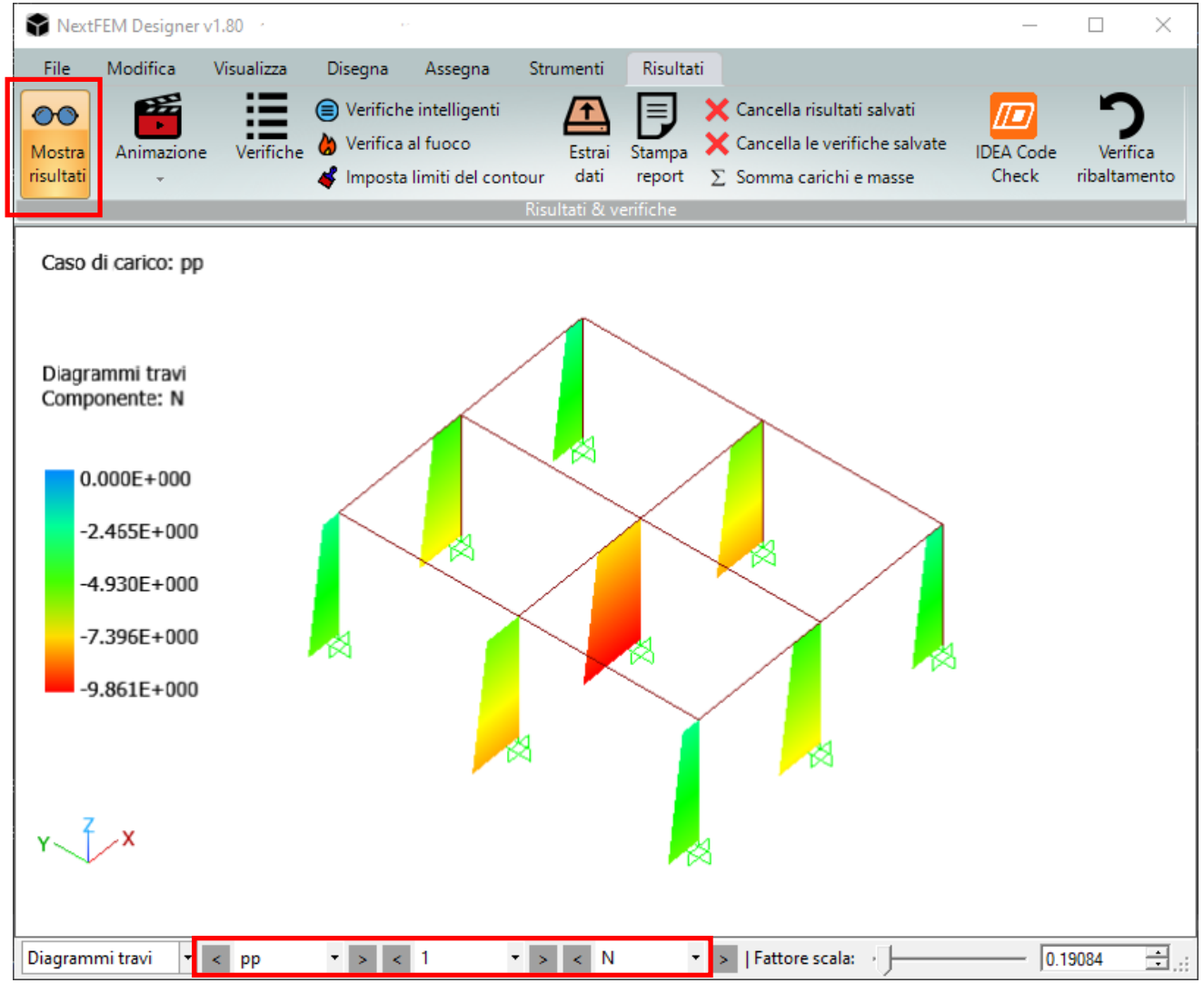
Import from CDM DOLMEN

3. Once the model has been imported, it is possible to switch from the wire view to the extruded view using the ***Extrude*** command.



Import from CDM DOLMEN

4. The model will always be imported in *daN* and *cm*.



The cases and load combinations have been imported in full.

To view the read stress diagrams, scroll through the read cases and combinations with the "Case" menu and the type of stress with the "Component" menu.

Export from CMP and SARGON

From CMP - Namirial

Use the command *File/Export/Export CMP template* for *CMP (Namirial).*
This will result in .SR3 and .SDB files.

Not all sections are supported, in particular:

- composed of several profiles (Composed),
- double UPN profiles,
- double angles not coupled to the back,
- omega profiles,
- quadruple angles.

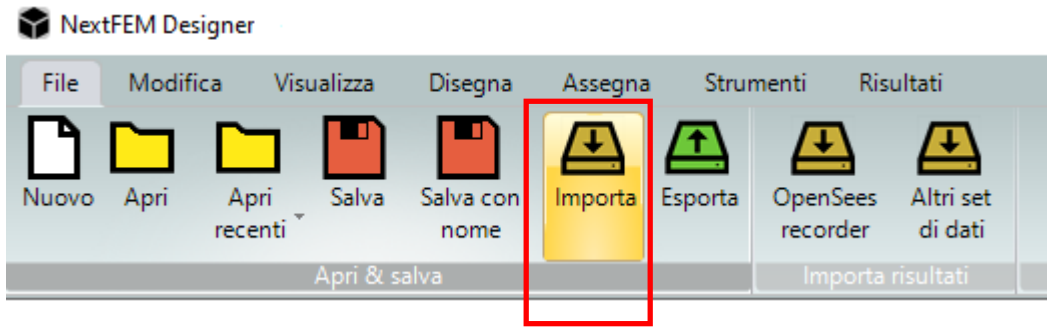
From SARGON - Castalia

Use the *File/Export* command for *Sargon (Castalia)*.
In the *File type* box, select ".sr3 (Sargon)" and click *OK*.
This will result in .SR3 and .SDB files.

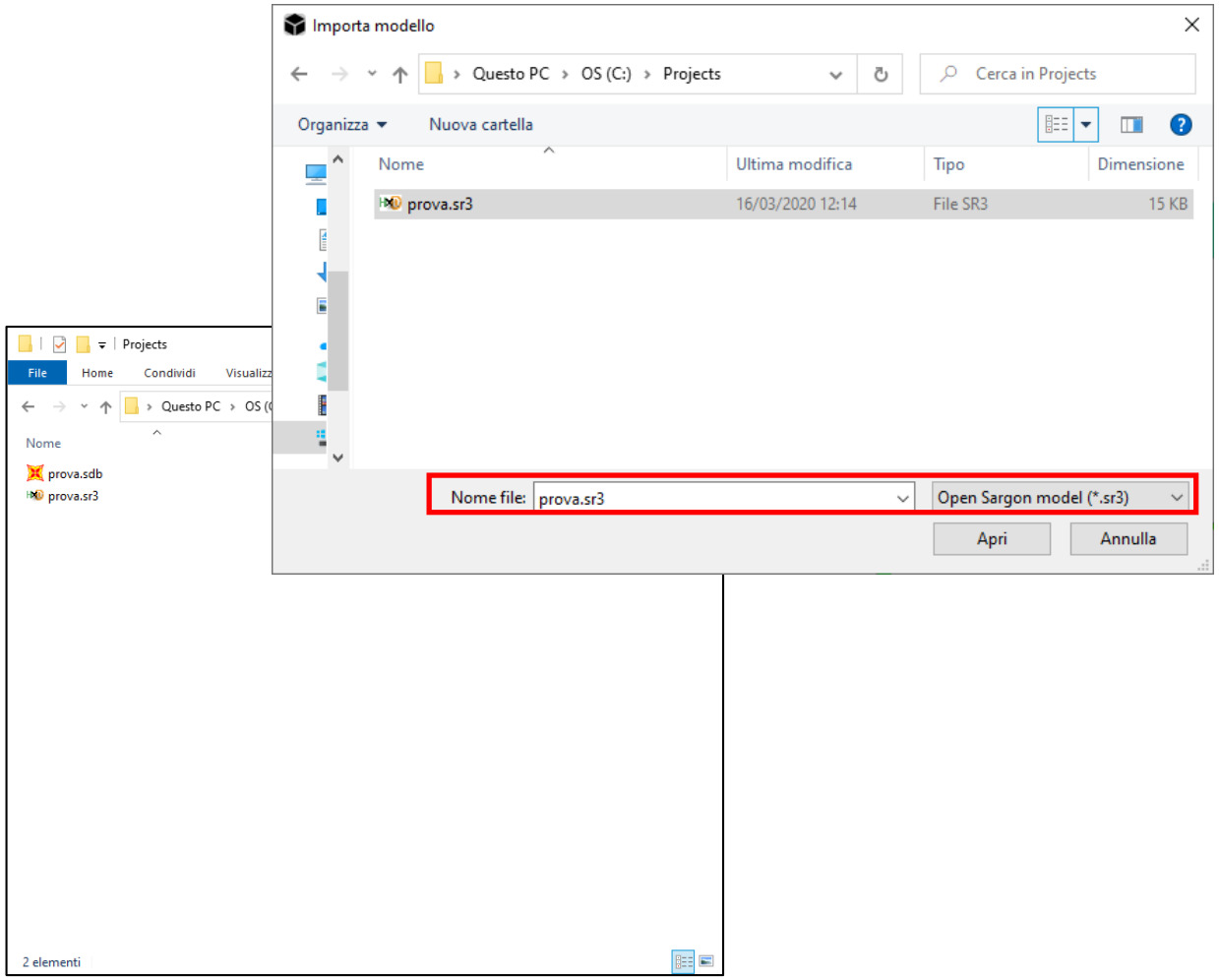
Rod materials are automatically identified if they are in NextFEM Designer's internal database.

Import from CMP and SARGON

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu

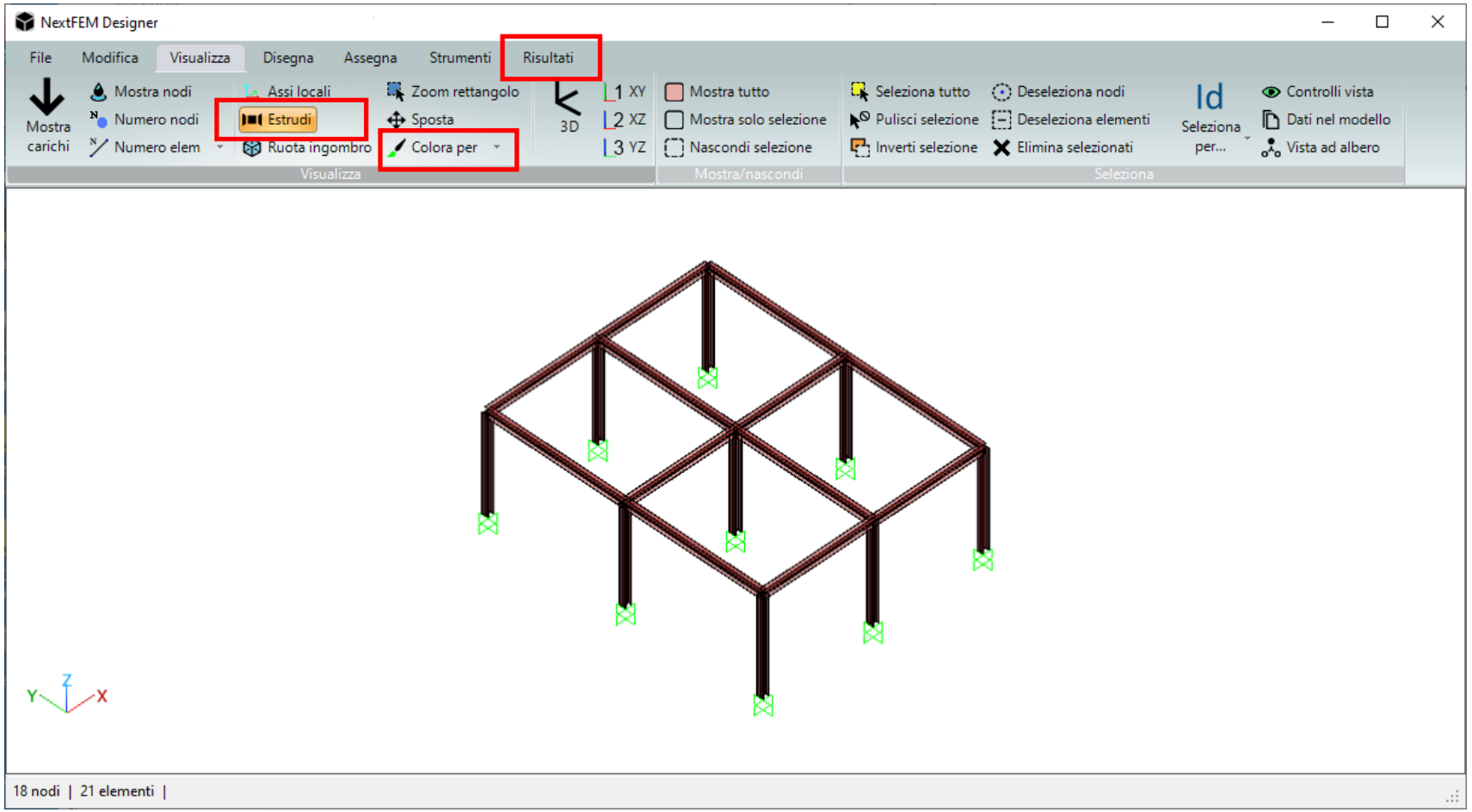


2. Select *Open Sargon model (*.sr3)* as the file type from the drop-down list or drag the model into the viewport.



Import from CMP and SARGON

3. Once the model has been imported, it is possible to switch from the wire view to the extruded view using the ***Extrude*** command.

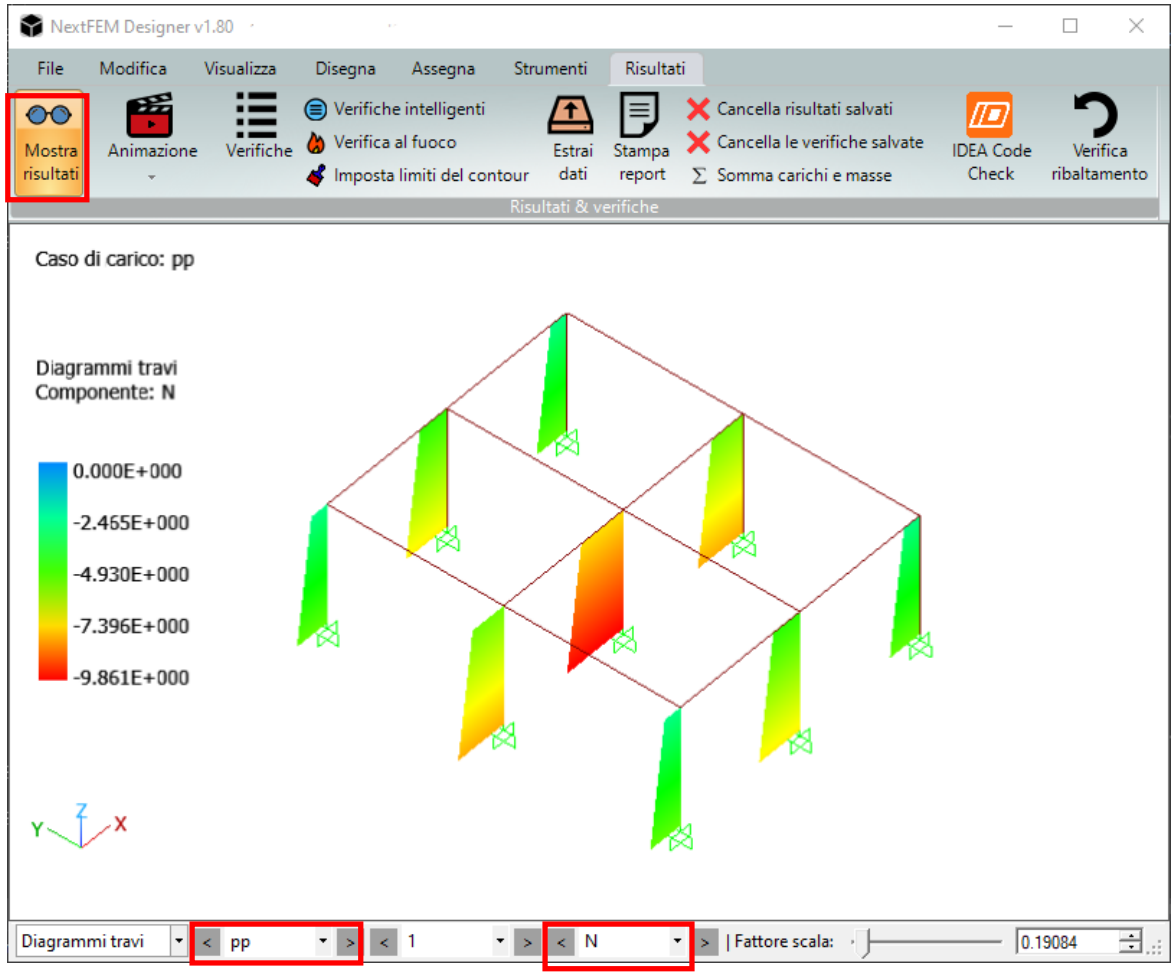


Use the "**Colour by**" command to colour each section differently.

The commands only take effect if the **Results** / "**Show results**" command is not active.

Import from CMP and SARGON

4. The model will always be imported in *N* and *mm*.



Cases and load combinations are referred to by the "short name" adopted within the CMP.

To view the read stress diagrams, scroll through the read cases and combinations with the "**Case**" menu and the type of stress with the "**Component**" menu.

Export from Midas GEN

1. In Midas GEN save the **MGT text file**, using the command
File / Export / Midas Gen MGT file ...

*This guide is also valid for Midas **Civil** - the format of the file exported by Midas Civil is .MCT.*

steel_building.mgt - Notepad++

File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?

steel_building.mgt

```
;------------------------------------------------------------------------------
;  midas Gen Text(MGT) File.
;  Date : 2020/1/15
;------------------------------------------------------------------------------

*VERSION
   8.2.0

*UNIT    ; Unit System
; FORCE, LENGTH, HEAT, TEMPER
   KN   , M, BTU, C

*STRUCTYPE    ; Structure Type
; iSTYP, iMASS, iSMAS, bMASSOFFSET, bSELFWEIGHT, GRAV, TEMPER, bALIGNBEAM, bALIGNSLAB, bROTRIGID
     0, 1, 1, NO, YES, 9806, 0, NO, NO, NO

*REBAR-MATL-CODE    ; Rebar Material Code
; CONC_CODE, CONC_MDB, SRC_CODE, SRC_MDB
 EN04(RC), Class A, ASTM(RC), Grade 60

*NODE    ; Nodes
; iNO, X, Y, Z
    22, 8, 15, 0
    23, 8, 20, 0
    24, 8, 25, 0
    31, 12, 15, 0
    32, 12, 20, 0
    33, 12, 25, 0
```

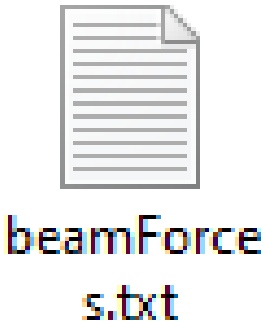
Normal text file | length : 23 923 | lines : 421 | Ln : 1 | Col : 1 | Sel : 0 | 0 | Unix (LF) | UTF-8 | INS

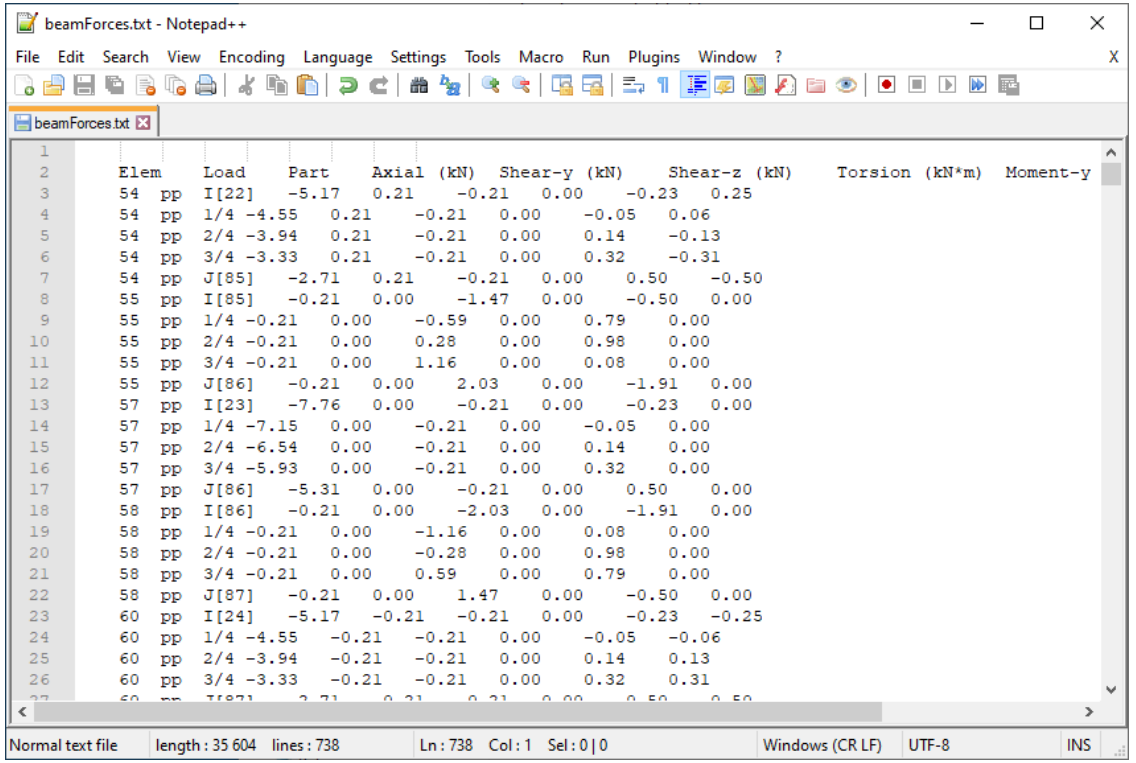
Export from Midas GEN

NextFEM Designer can import results related to:

- **Stress on beams** (export from GEN with *Results / Result tables / Beam / Force*)
- **Rod stresses** (export from GEN with *Results / Result tables / Truss / Force)*
- **Stress on septa** (export from GEN with *Results / Result tables / Wall / Force/moment)*
- **Nodal displacements** (export from GEN with *Results / Result tables / Displacements)*
- **Elastic link stresses (**export from GEN with *Results / Results tables / Elastic Link*)

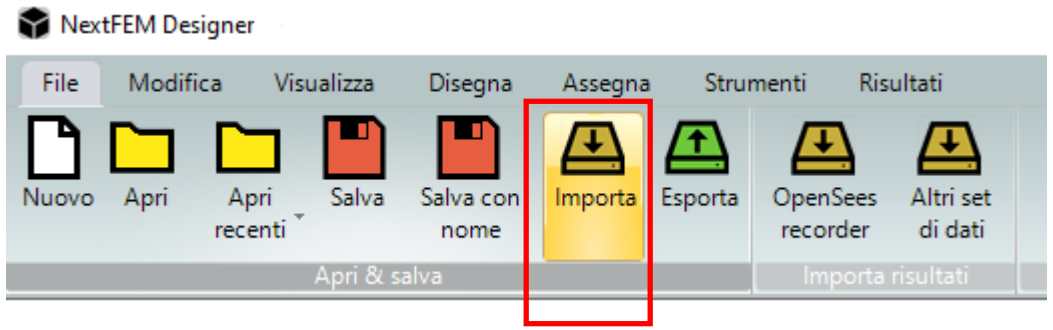
2. Paste the tables of results into a text file (. txt, . dat etc.).



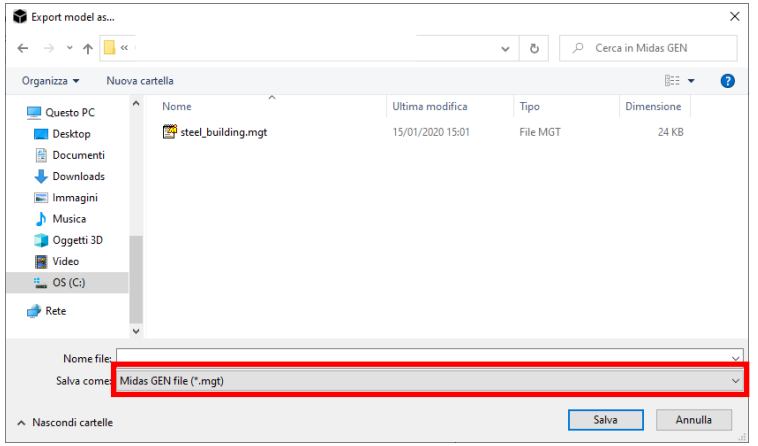


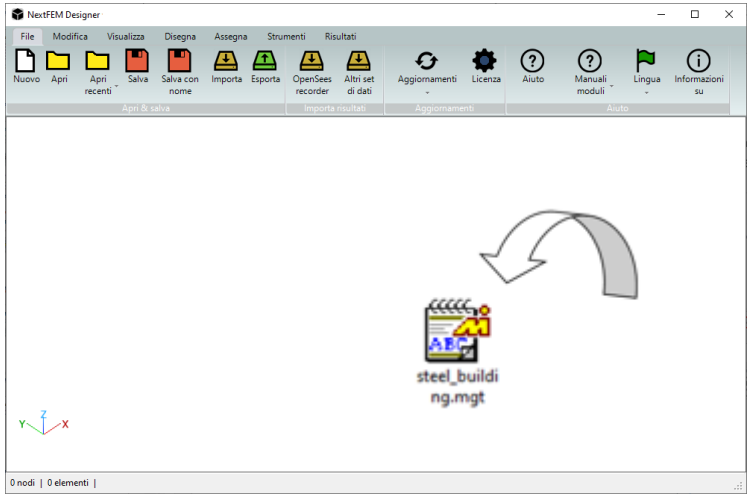
Importing from Midas GEN and Civil

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu



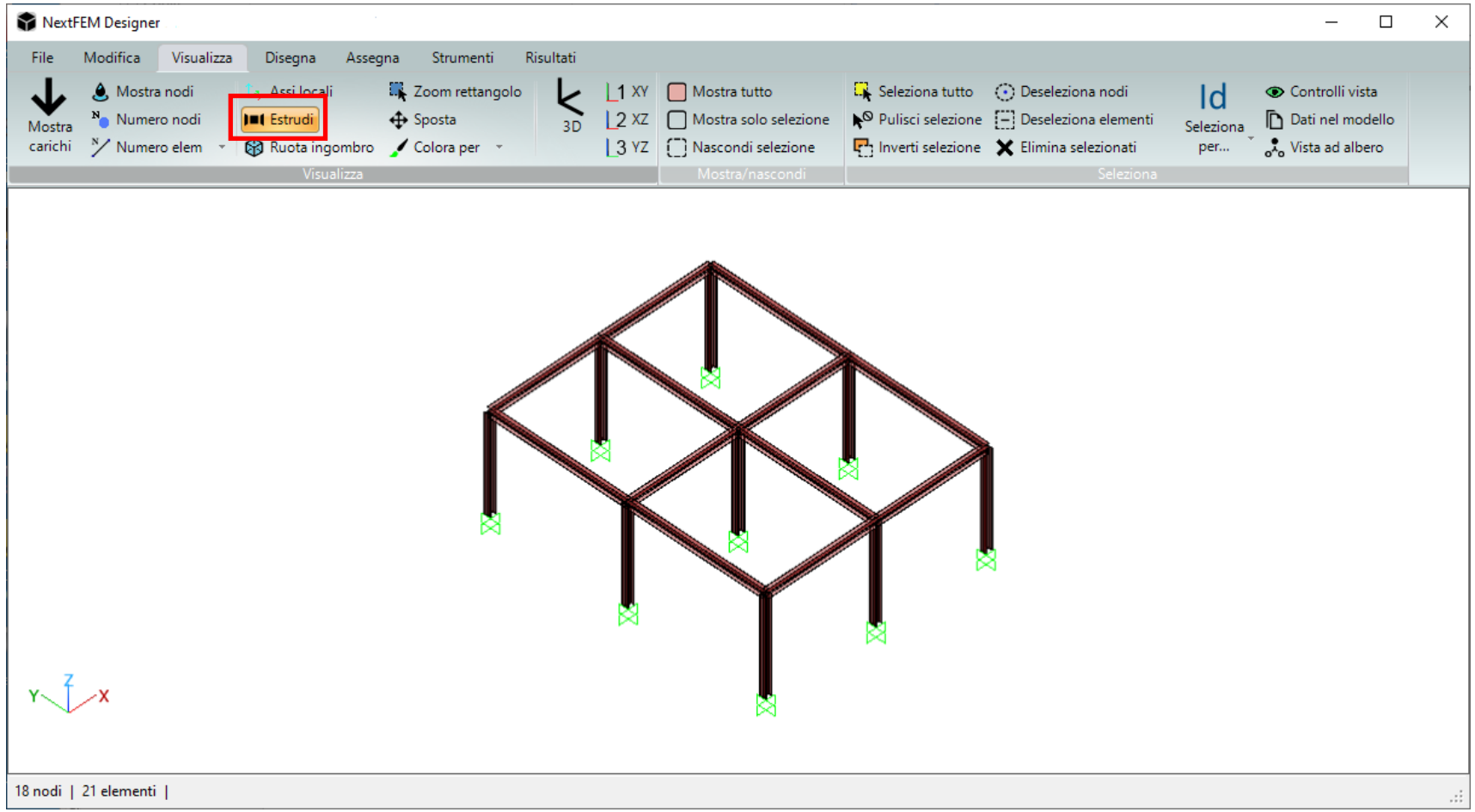
2. Select *Midas GEN text file (*.mgt)* as the file type from the drop-down list, or drag the MGT/MCT model into the viewport.





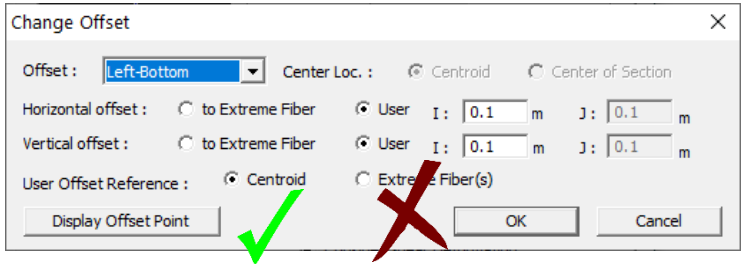
Importing from Midas GEN and Civil

3. Once the model has been imported, it is possible to switch from the wire view to the extruded view using the *Extrude* command.



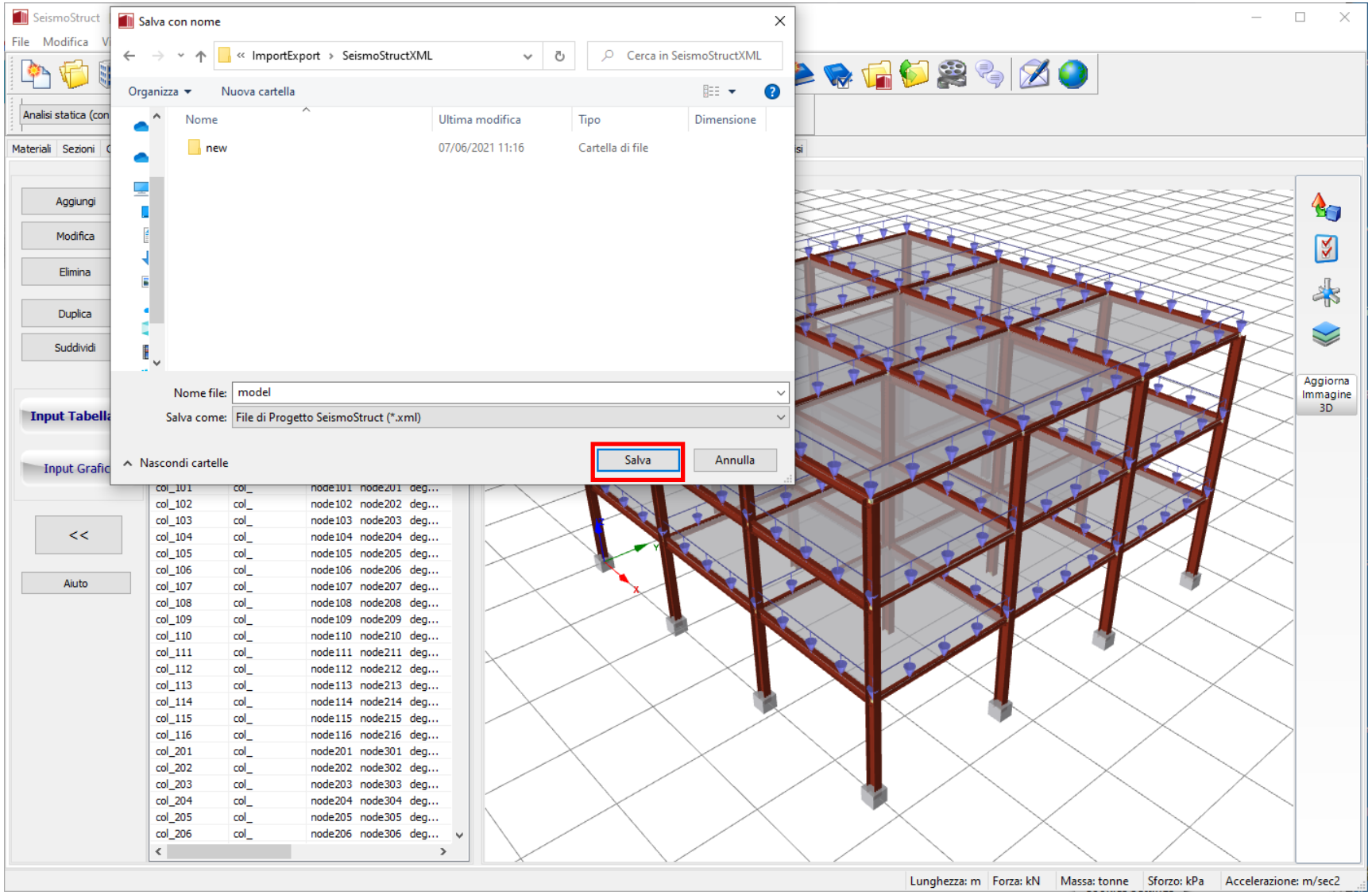
NOTE:

The import supports any section offsets, which must always be defined in GEN from the centre of the section, and NOT from the extreme fibres.



Export from SeismoStruct

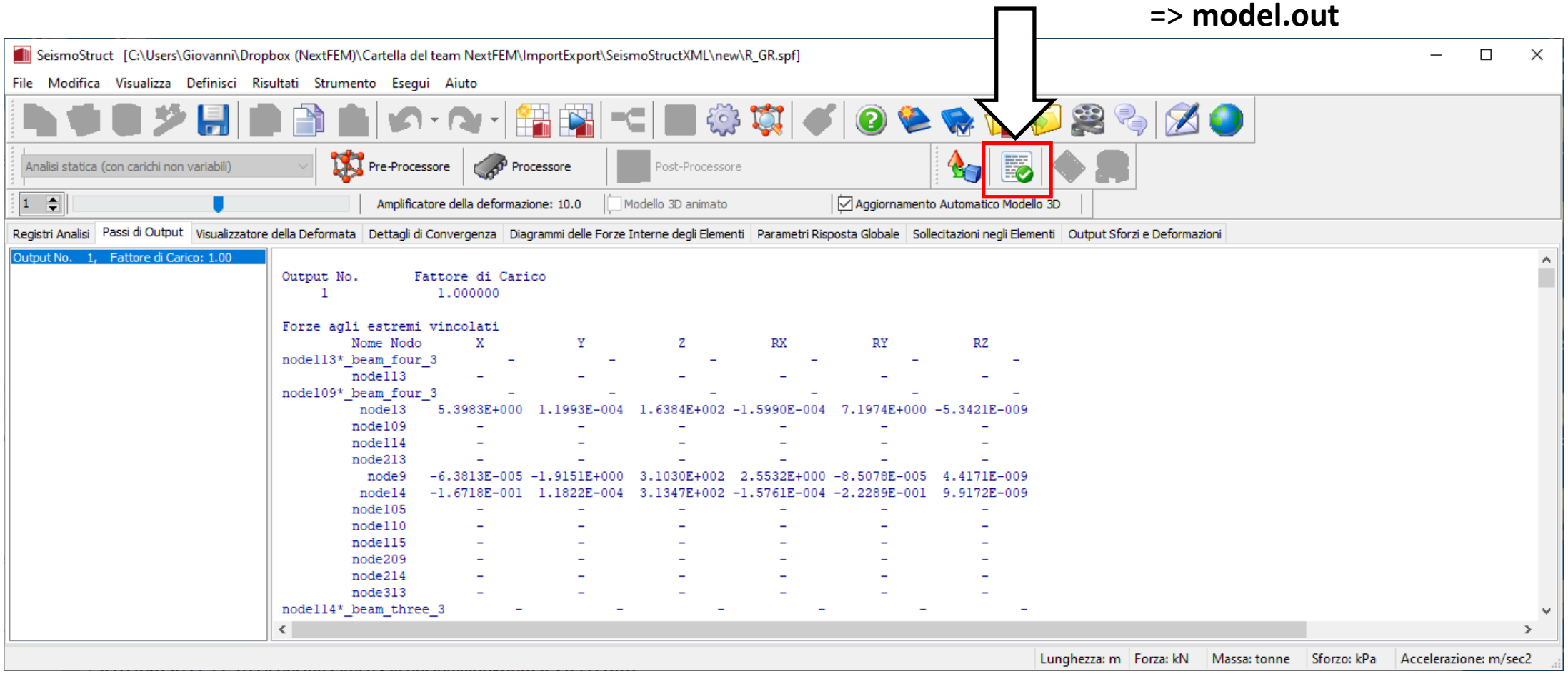
1. Within the SeismoStruct environment, access the command *File / Export as XML...*
   Then save the file in the chosen folder.



Export from SeismoStruct

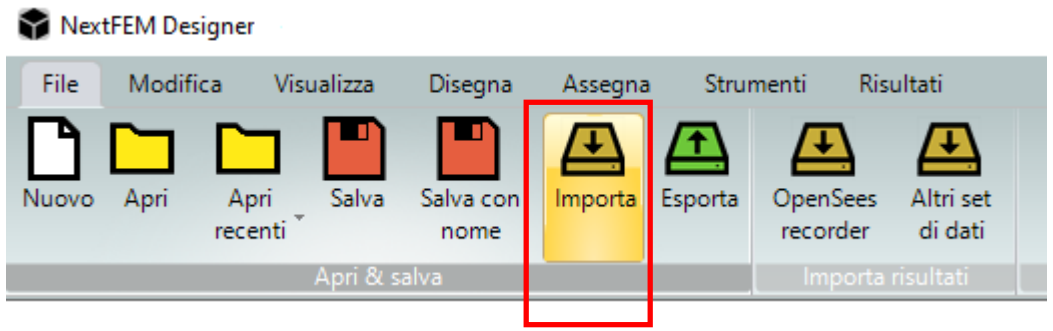
2. It is necessary to export the results of the analysis to an .out file with the command highlighted in the figure below, which can be accessed when the programme is in *Post-Processor* mode.

N.B. The .out file must be saved with the same name as the XML file.
E.g. **model.xml**
=> **model.out**

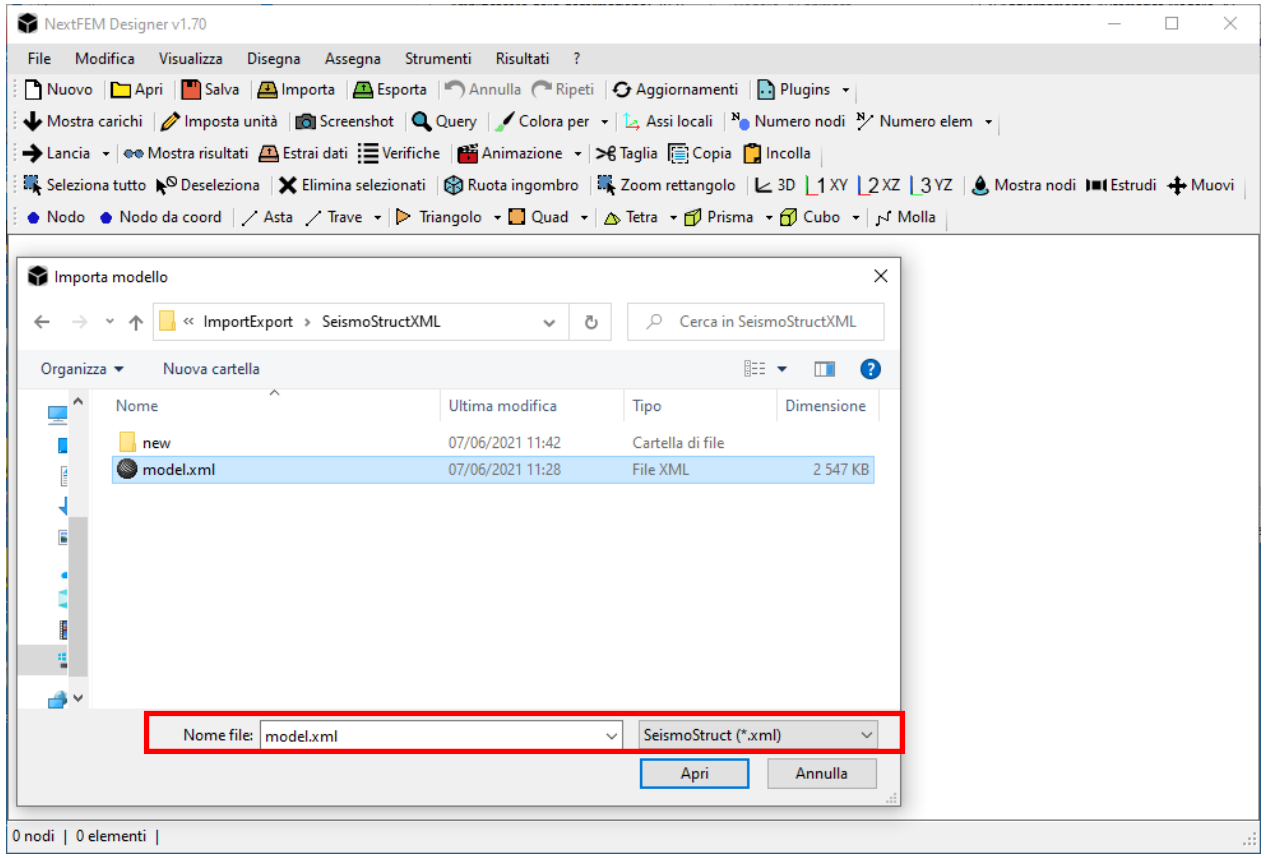


Import from SeismoStruct

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu.

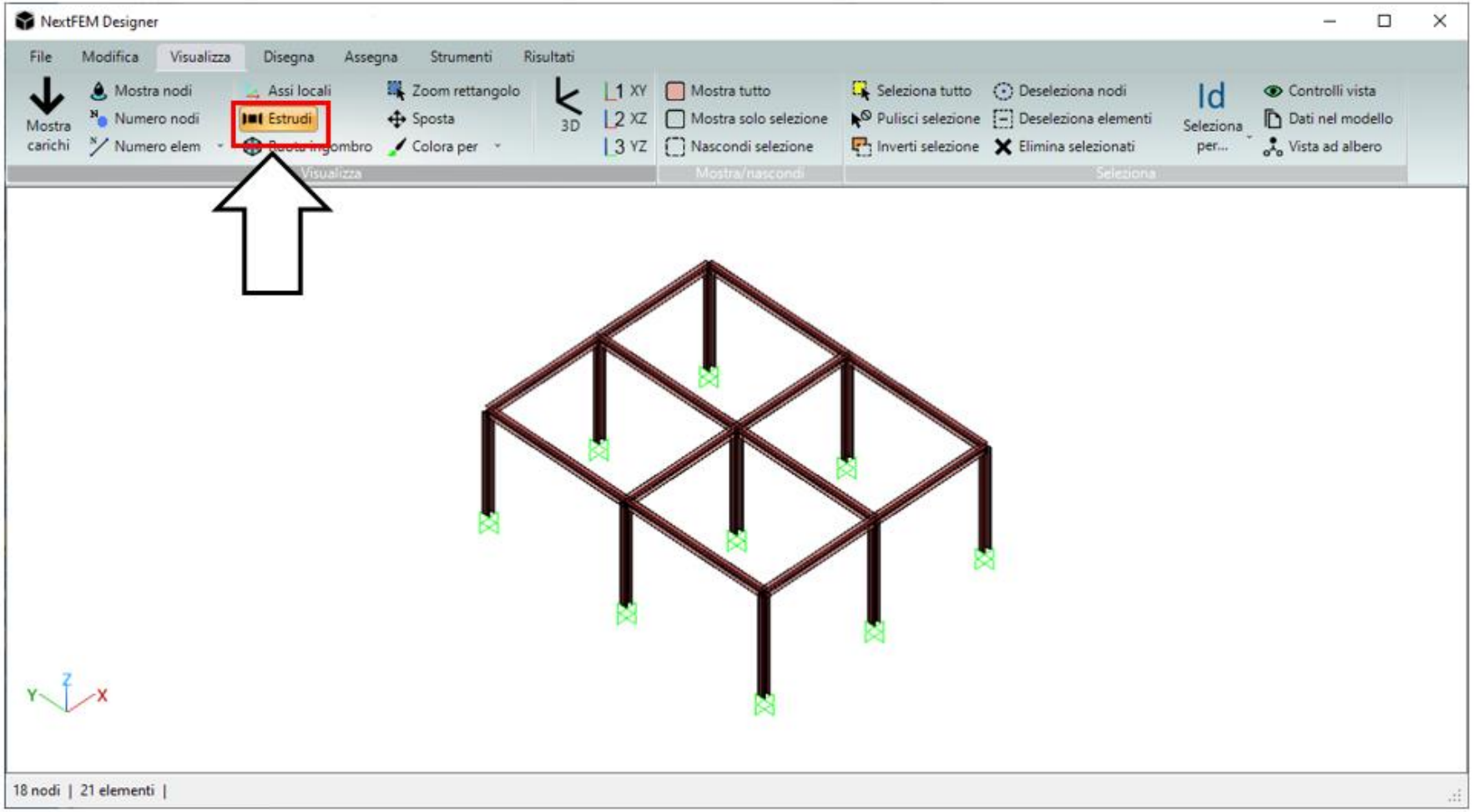


2. Select *SeismoStruct (*.xml)* as the file type from the drop-down list, or drag the model into the viewport.



Import from SeismoStruct

3. Once the model has been imported, you can switch from wireframe view to extruded view using the *Extrude* command. This option only works if *Show results* is unchecked.

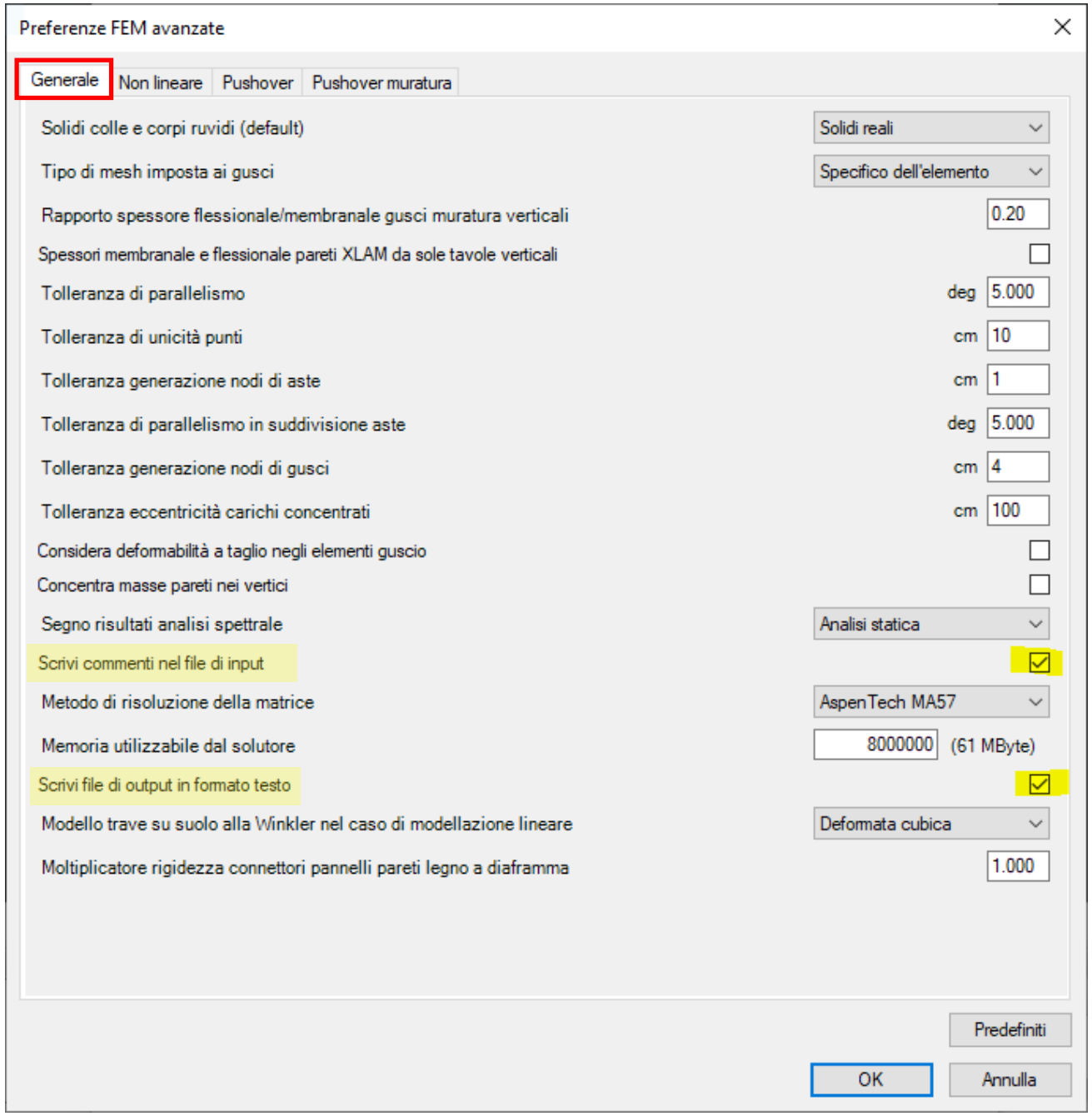


Export from Sismicad

1. Within the SismiCAD environment, go to the *Database / Preferences ...* mask, *FEM* tab, and click on *Advanced*. In this mask, activate the options:

✓ Write comments in the input file;
✓ Write output files in text format.

2. Then launch the *Tools/FEM Calculation* command.



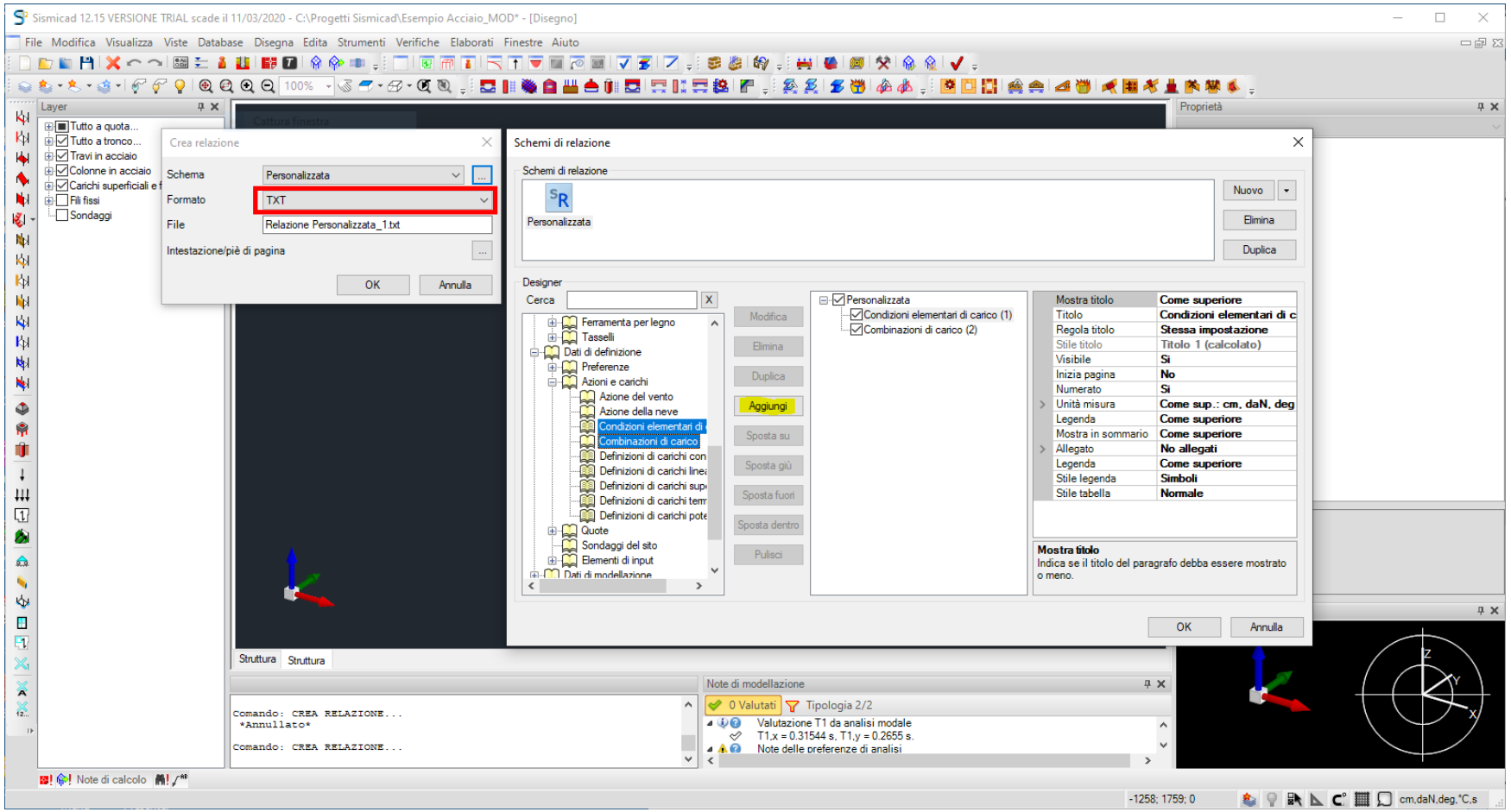
Export from Sismicad

Sections and materials are not read. They are assigned by default:
- the material of all imported elements is **S235 (*)**;
- the cross section of all imported elements is **IPE80 (*)**.

Only FRAME and TRUSS, i.e. beams and connecting rods, are supported.

(*) Sections and rod materials are automatically identified if they are in Designer's internal database.

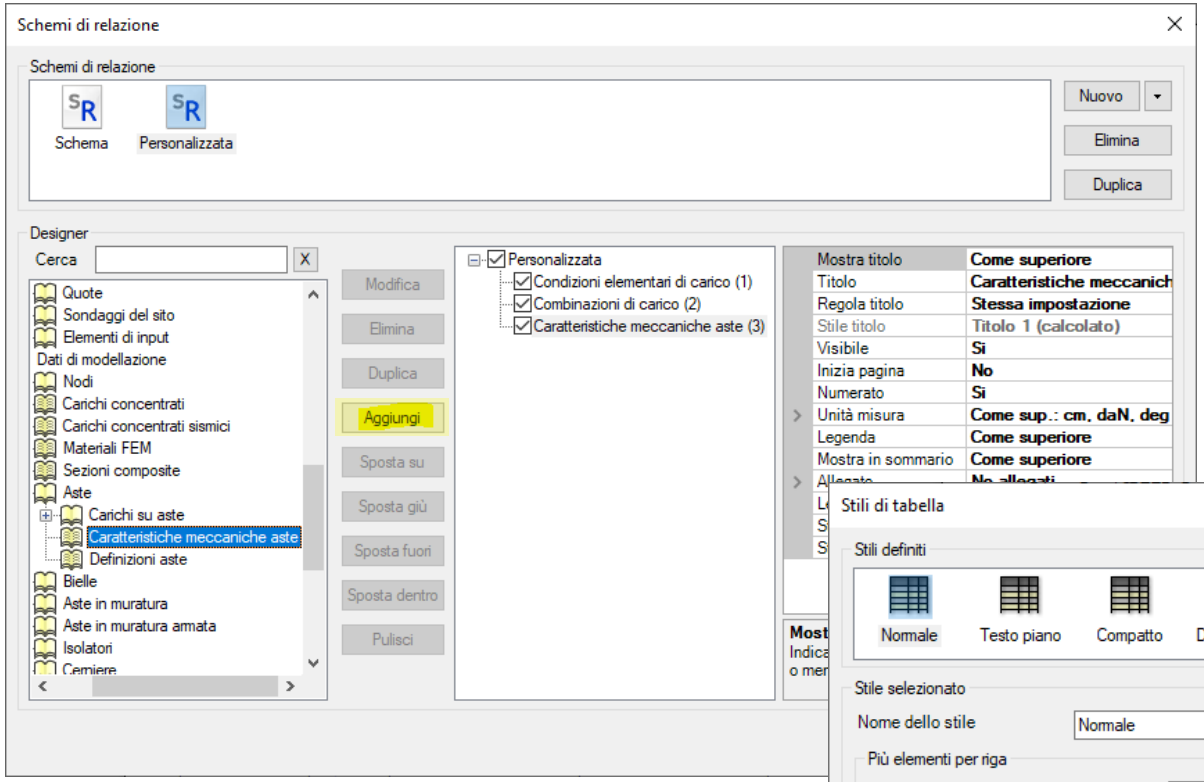
Before importing, it is necessary to export the load combinations in TXT format from the command ***Processes / Reports / Create report***.

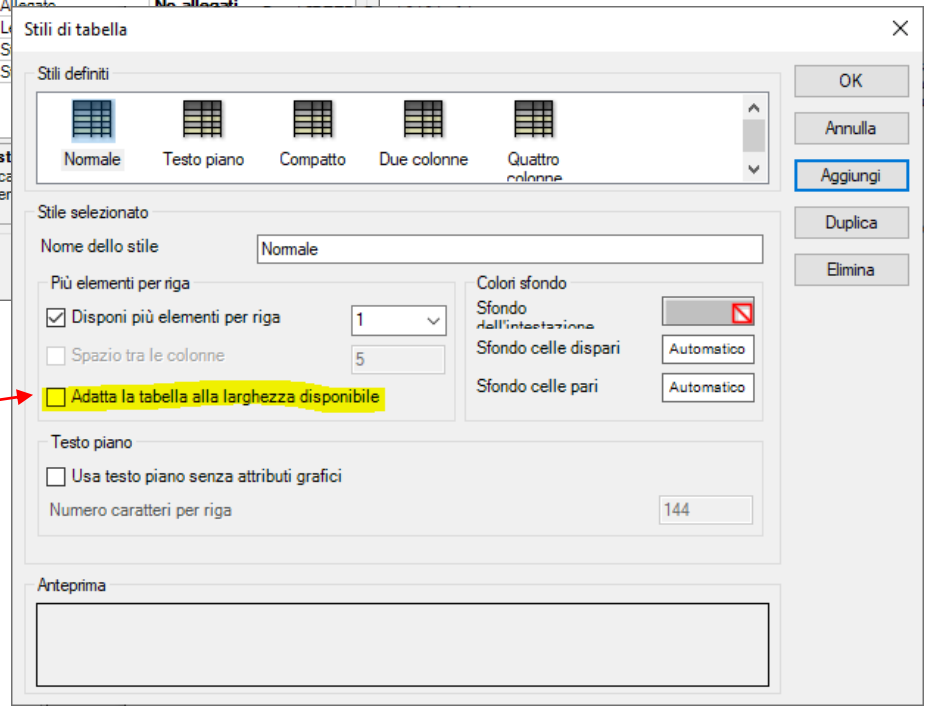


Export from Sismicad

Press the "..." button and add the following items to the "**Custom**" format, following the order:

- ✓ Definition data / Actions and loads / **Basic load conditions**
- ✓ Definition data / Actions and loads / **Load combinations**
- ✓ Modelling data / Rods / **Mechanical properties of rods**.



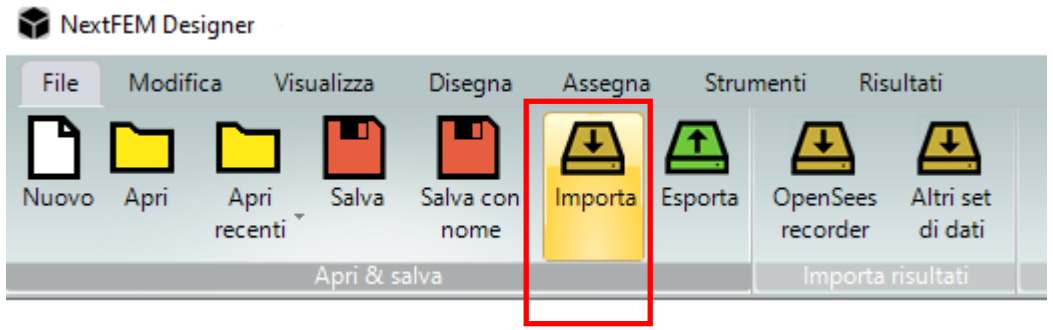


In Processes / Reports / Table styles, deactivate "Fit table to available width" for the "Normal" style.

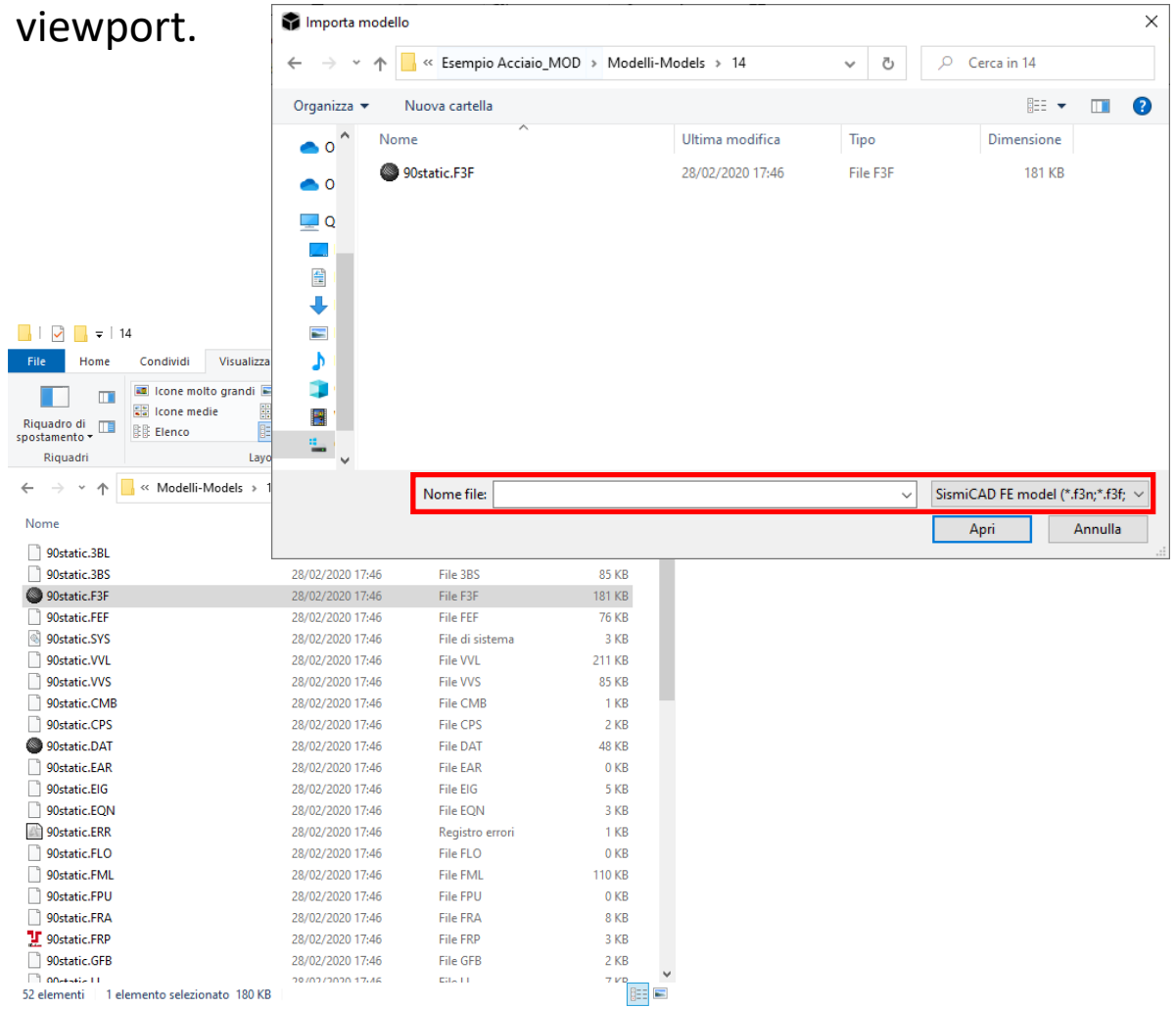
Generate the relationship.

Importing from Sismicad

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu

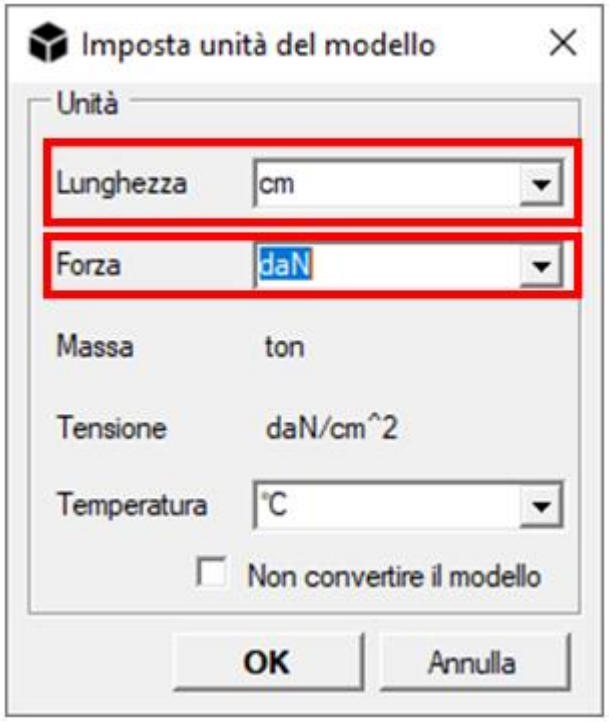


2. Select *Sismicad FE model (*.f3f, .f3n, ...)* as the file type from the drop-down list or drag the model into the viewport.

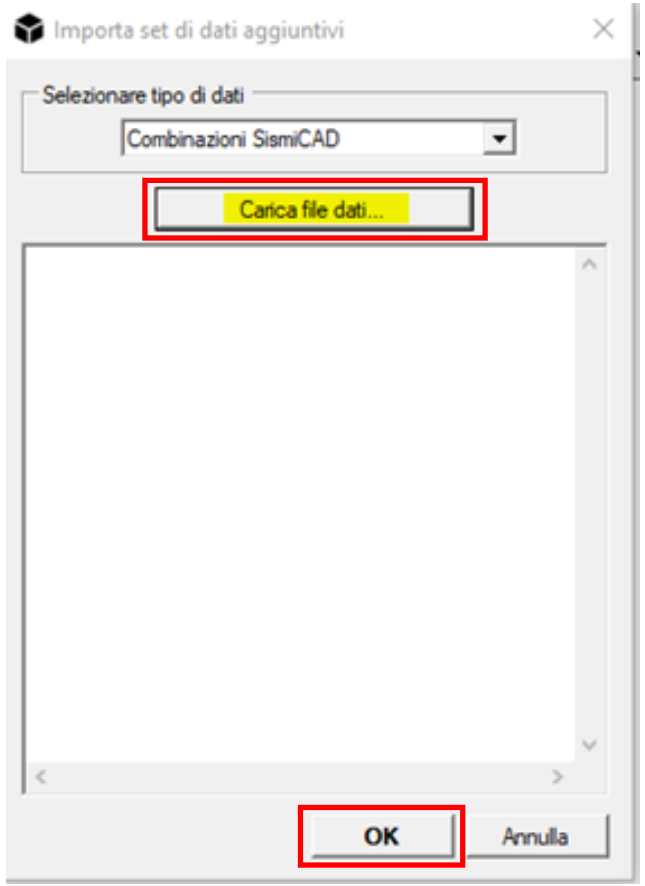


Importing from Sismicad

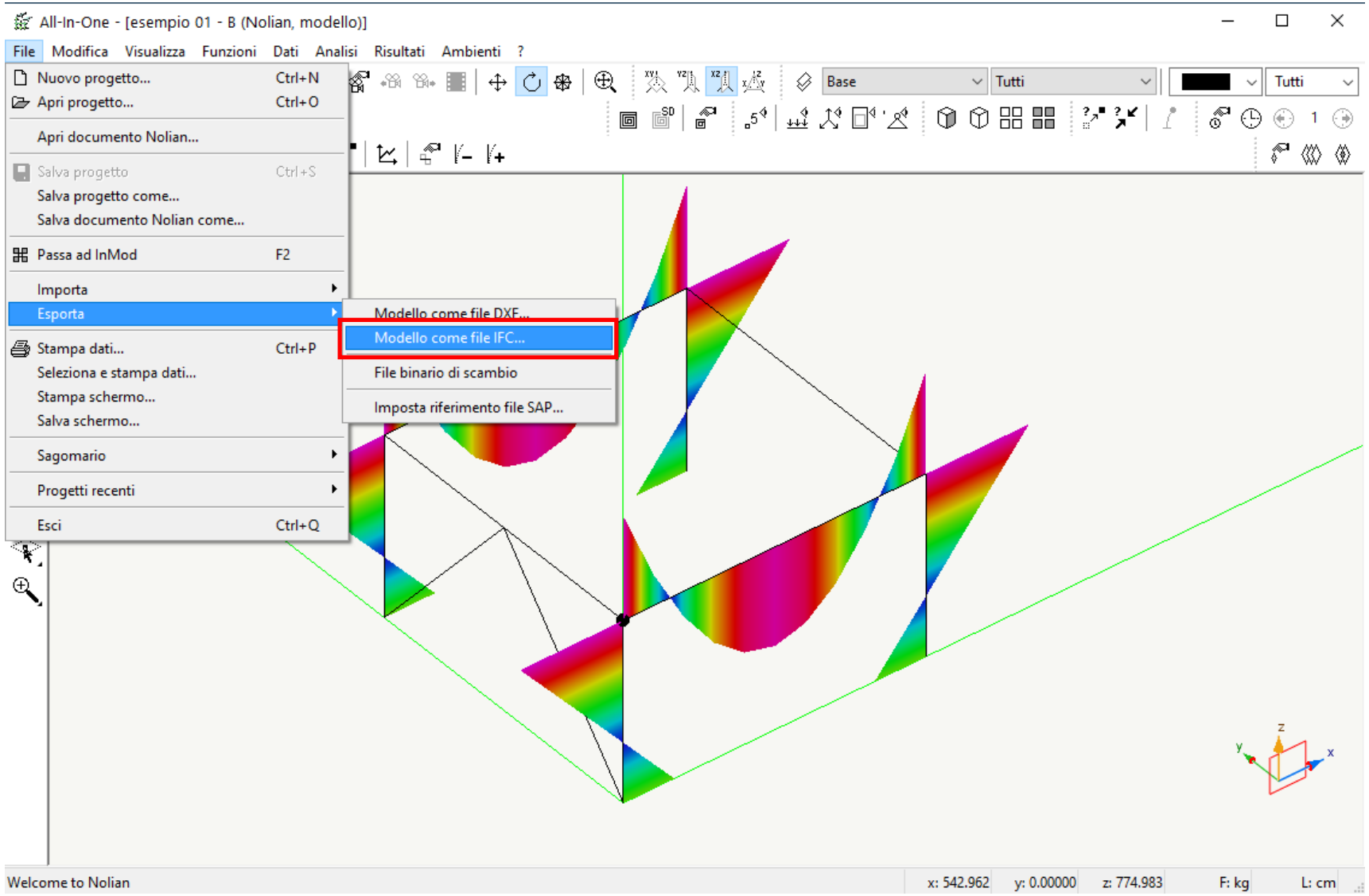
3. The units window appears: set the units used in SismiCAD for *Length* and *Force*. Give *OK*.



4. The load combinations previously saved in the report in TXT are required:
Click the *Upload data file...* button to navigate to the job folder and select the report saved as a TXT in the *Print-Prints* subfolder.
Give *OK*.



Export from Softing Nolian



To export the model from Nolian once the analysis has been performed, select the command:

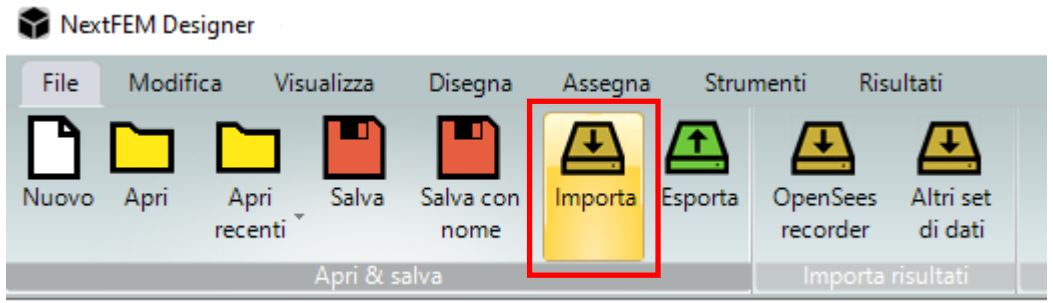
File / Export / Model as IFC file ...

The command produces a file with an **IFC** extension

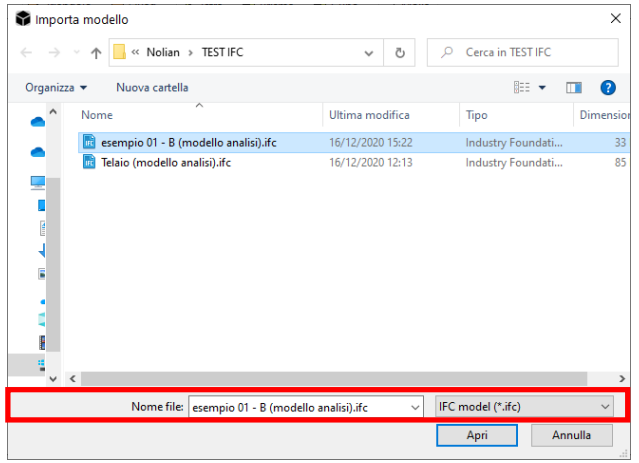
esempio 01 - B (modello analisi).ifc

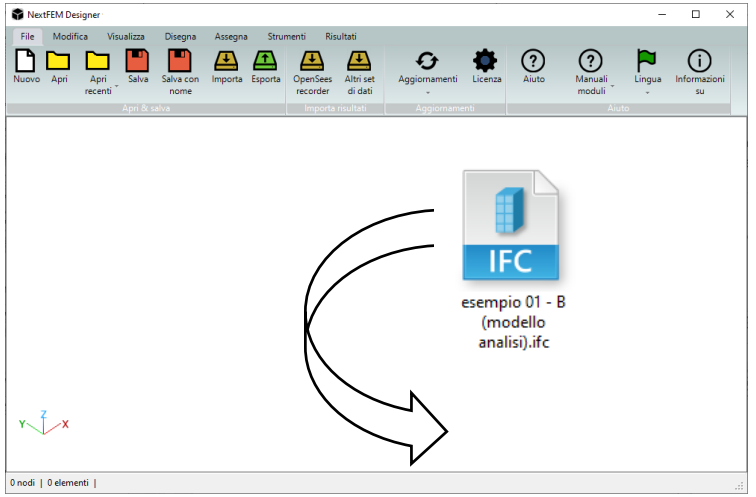
Imported by Softing Nolian

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu



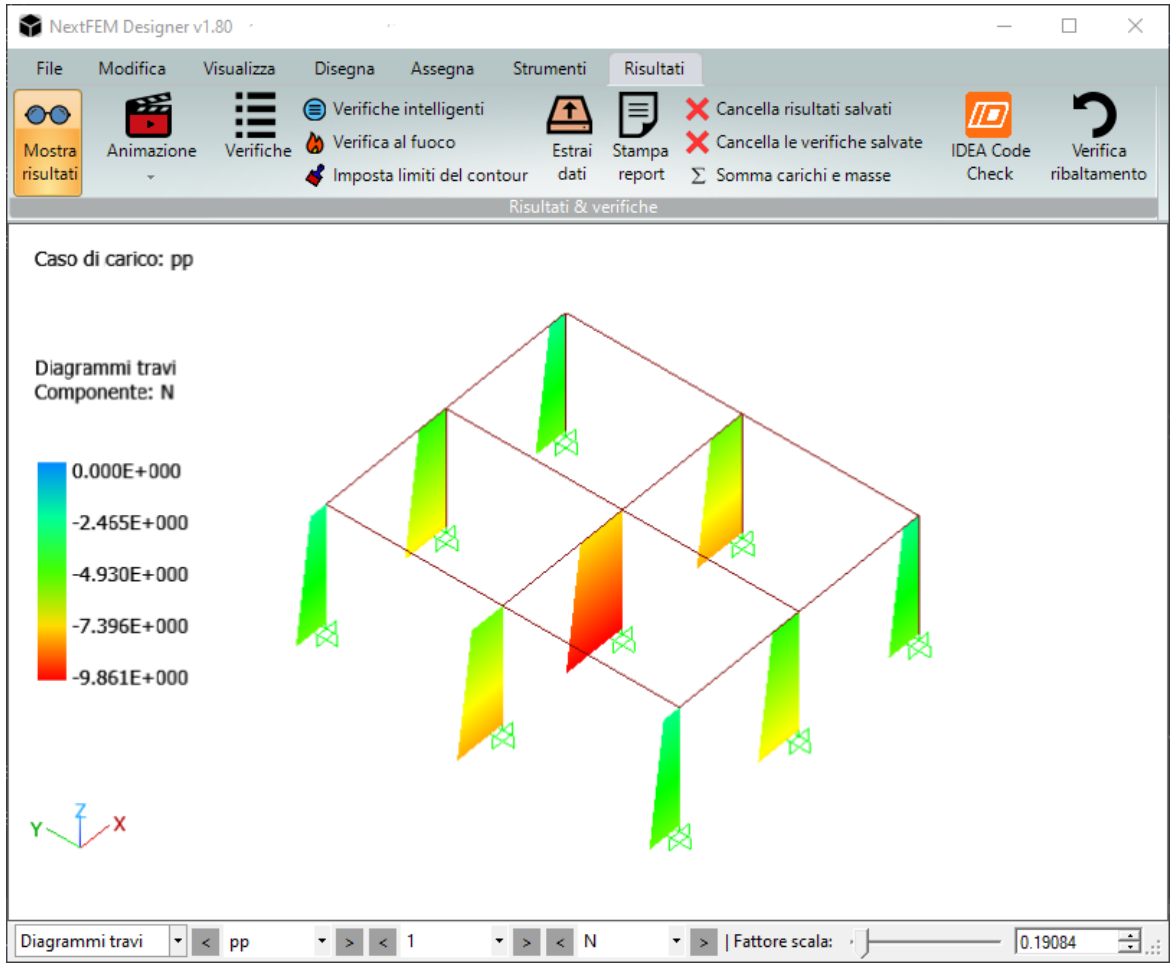
2. Select *IFC model (*.ifc)* as the file type from the drop-down list or drag the IFC model into the viewport.





Imported by Softing Nolian

3. Once the model has been imported, the stresses of the load combinations/conditions read are displayed.



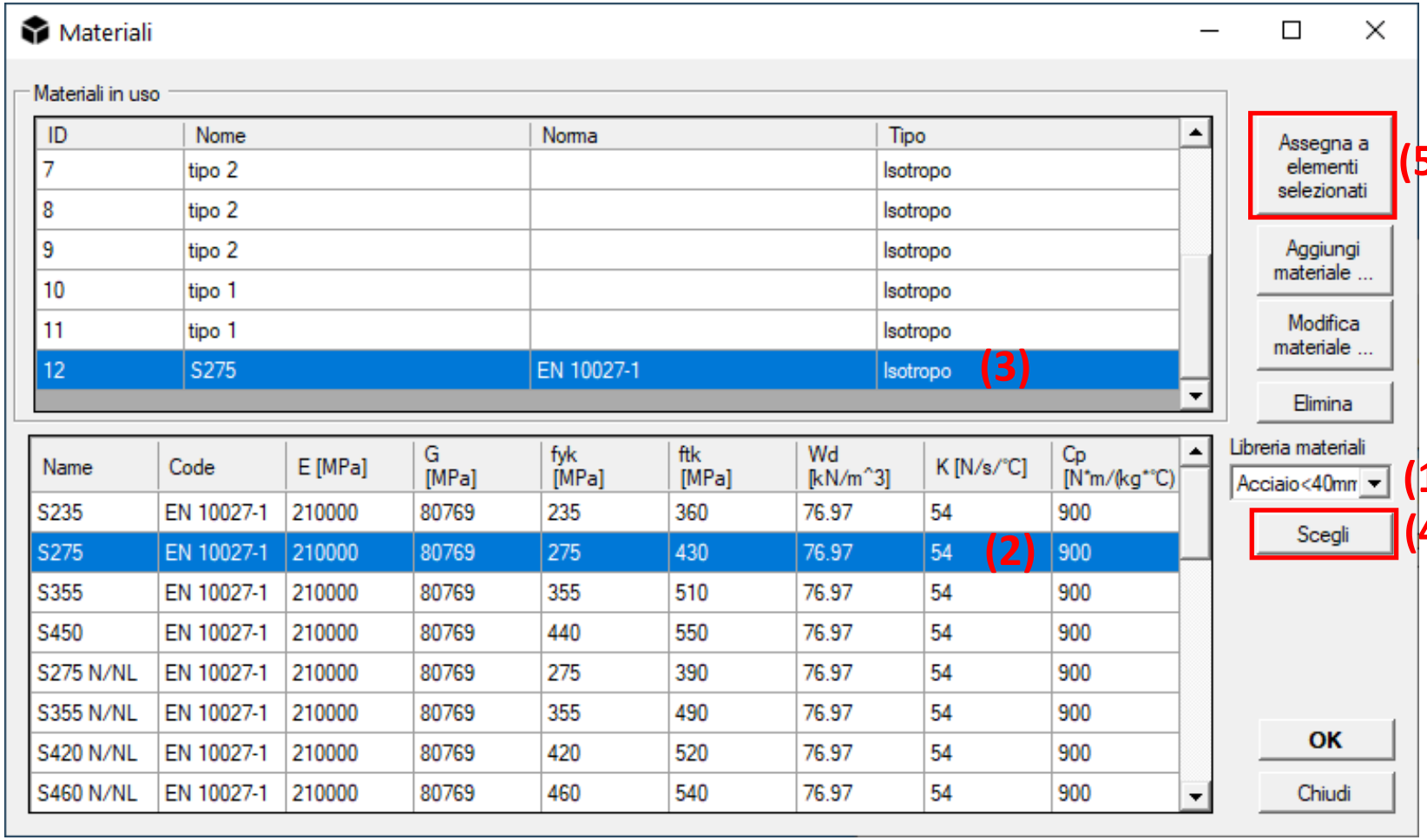
NOTE:

If the stresses are not displayed immediately, check to:

- ✓ Having performed the analysis in Nòlian before exporting
- ✓ You have activated your NextFEM Designer licence.

Imported by Softing Nolian

4. It is generally required to reassign the steel used, using the *Edit / Materials* command.



(1)From the material library select 'Steel<40mm', (2)-(3)select a row from the box below (e.g. S275) and press (4)*Choose*.

Select the chosen steel and the whole structure in the viewport (hold down the left mouse button and drag the selection box as in CAD).

(5)Finally, press "*Assign to selected items*".

Export from Straus7

1. In Straus7, select the *File / Export* command and choose *Straus7 text file* as type.
A text file containing the template is obtained.

Mesh3.txt - Blocco note di Windows

File Modifica Formato Visualizza ?

```
/ ____________________________________________________________________
/ COORDINATE SYSTEM DEFINITIONS

   CoordSys                1   "Global XYZ"  GlobalXYZ

   CoordSys                2   "UCS 1"  RectUCS
    1.50000000000000E+3      1.30000000000000E+4      1.50000000000000E+3
    3.00000000000000E+3      1.25000000000000E+4      2.50000000000000E+3
    0.00000000000000E+0      1.25000000000000E+4      2.50000000000000E+3


/ ____________________________________________________________________
/ NODE COORDINATES

   Node                    1     0.00000000000000E+0      0.00000000000000E+0      0.00000000000000E+0
   Node                    2     3.00000000000000E+3      0.00000000000000E+0      0.00000000000000E+0
   Node                    3     0.00000000000000E+0      0.00000000000000E+0      2.50000000000000E+3
   Node                    4     3.00000000000000E+3      0.00000000000000E+0      2.50000000000000E+3
   Node                    5     1.50000000000000E+3      5.00000000000000E+2      1.50000000000000E+3
   Node                    6     0.00000000000000E+0      2.50000000000000E+3      2.50000000000000E+3
   Node                    7     0.00000000000000E+0      5.00000000000000E+3      2.50000000000000E+3
   Node                    8     0.00000000000000E+0      7.50000000000000E+3      2.50000000000000E+3
   Node                    9     0.00000000000000E+0      1.00000000000000E+4      2.50000000000000E+3
   Node                   10     0.00000000000000E+0      1.25000000000000E+4      2.50000000000000E+3
   Node                   11     3.00000000000000E+3      2.50000000000000E+3      2.50000000000000E+3
   Node                   12     3.00000000000000E+3      5.00000000000000E+3      2.50000000000000E+3
   Node                   13     3.00000000000000E+3      7.50000000000000E+3      2.50000000000000E+3
   Node                   14     3.00000000000000E+3      1.00000000000000E+4      2.50000000000000E+3
   Node                   15     3.00000000000000E+3      1.25000000000000E+4      2.50000000000000E+3
   Node                   16     1.50000000000000E+3      3.00000000000000E+3      1.50000000000000E+3
   Node                   17     1.50000000000000E+3      5.50000000000000E+3      1.50000000000000E+3
   Node                   18     1.50000000000000E+3      8.00000000000000E+3      1.50000000000000E+3
   Node                   19     1.50000000000000E+3      1.05000000000000E+4      1.50000000000000E+3
```

Linea 1, colonna 1 | 100% | Windows (CRLF) | UTF-8

Export from Straus7

The programme can import results related to:

- Stress on beams
- Nodal shifts

NextFEM Designer imports the tables available in Straus7 with the command *Results / Listings*.

beamForces.txt

Select *Multiple cases* in the results display window and make sure that *Column separator: TAB* is enabled by right-clicking on the table.

For the nodal displacement table, right-click on the table, click on the *Select All* command, then again click on *Copy* from the right-click menu *(+ Fixed Cells)*.

For the beam results table, set:

- *VIEW: Beam stations*
- *SORT: All beams*
- *SORT USING: Actual values*
- *FILTER ON: All beams*

Then right-click on the table, click on the *Select All* command, then again click on *Copy* from the right-click menu *(+ Fixed Cells)*.

*beamForces.txt - Notepad++

File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?

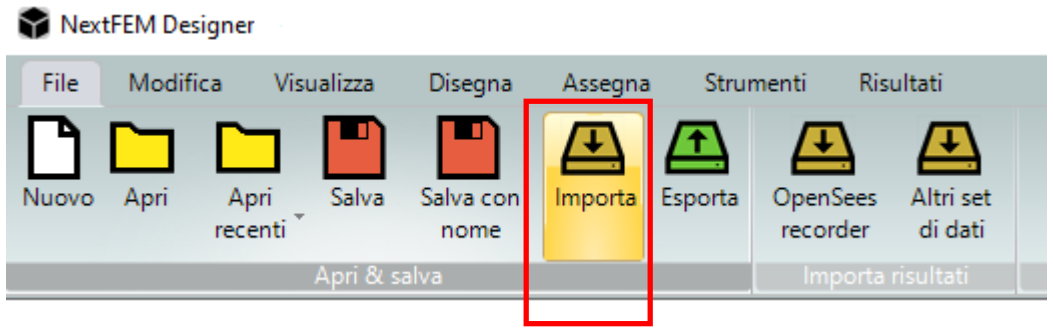
beamForces.txt

	Shear Force 1	Bending Moment 1	Shear Force 2	Bending Moment 2	Axial Force	Torque
	(N)	(N.mm)	(N)	(N.mm)	(N)	(N.mm)
Beam 1: 1: sw:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	-1154.1649	0.0000
Beam 1: 1: sw:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	-1154.1649	0.0000
Beam 1: 2: wx+:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 1: 2: wx+:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 1: 3: wy+:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 1: 3: wy+:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 1: 4: Seismic Case 1:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 1: 4: Seismic Case 1:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 1: 5: SLU1 [Combination 1]:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.4144	0.000
Beam 1: 5: SLU1 [Combination 1]:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.4144	0.000
Beam 1: 6: SLU2 [Combination 2]:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.4144	0.000
Beam 1: 6: SLU2 [Combination 2]:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.4144	0.000
Beam 1: 7: SLU3 [Combination 3]:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.4144	0.000
Beam 1: 7: SLU3 [Combination 3]:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.4144	0.000
Beam 1: 8: SEISM1 [Combination 4]:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.000
Beam 1: 8: SEISM1 [Combination 4]:	2500.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.000
Beam 2: 1: sw:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 2: 1: sw:	3000.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 2: 2: wx+:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.0000	0.0000
Beam 2: 2: wx+:	3000.0000 mm	0.0000	0.0000	0.0000	0.0000	-1500.0000	0.0000
Beam 2: 3: wy+:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 2: 3: wy+:	3000.0000 mm	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Beam 2: 4: Seismic Case 1:	0.0000 mm	0.0000	0.0000	0.0000	0.0000	-1310.8033	0.0000
Beam 2: 4: Seismic Case 1:	3000.0000 mm	0.0000	0.0000	0.0000	0.0000	-1310.8033	0.0000

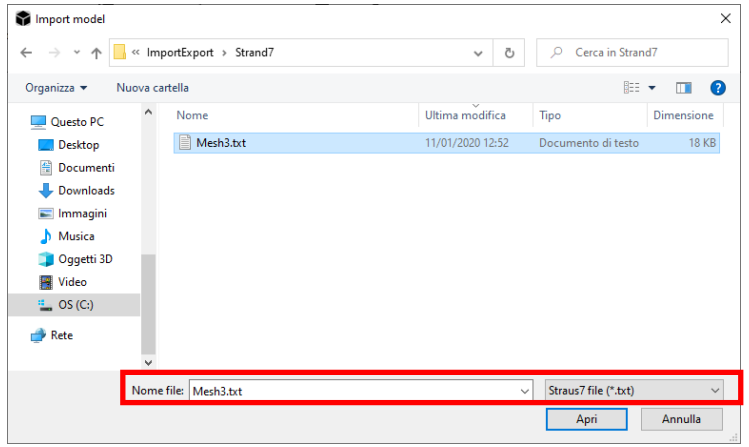
Normal text file | length : 51 972 lines : 601 | Ln : 1 Col : 1 Sel : 1 | 1 | Windows (CR LF) | UTF-8 | INS

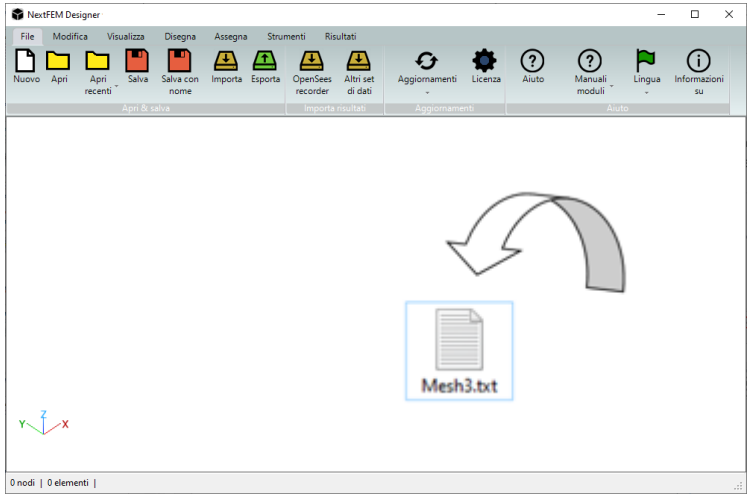
Importing from Straus7

1. In *NextFEM Designer,* choose the *Import* command from the *File* menu



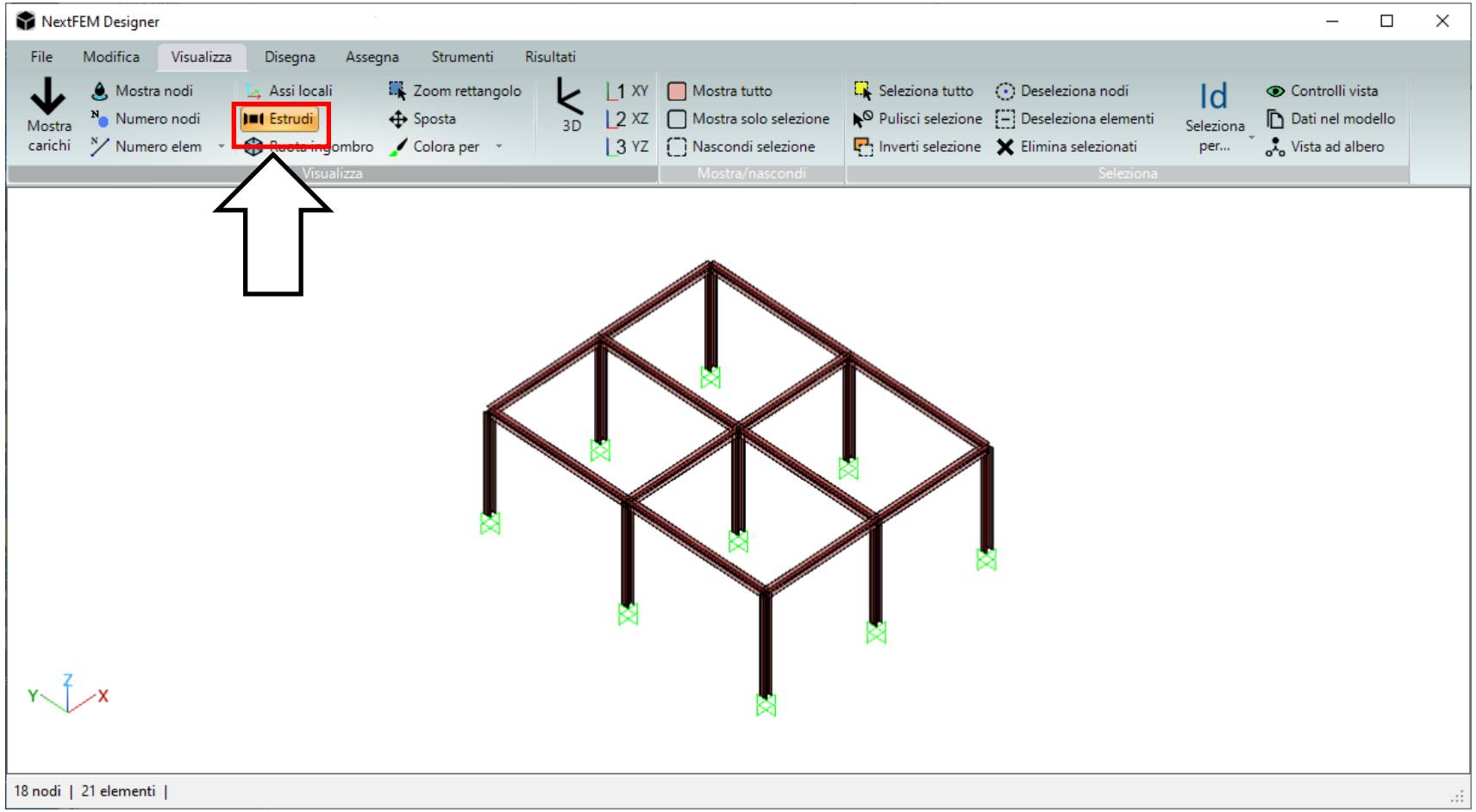
2. Select *Straus7 file (*.txt)* as the file type from the drop-down list or drag the model into the viewport.





Importing from Straus7

3. Once the model has been imported, it is possible to switch from the wire view to the extruded view using the *Extrude* command.



NOTE: for special needs concerning translation of Sofistik model, please contact us.

facebook.com/nextfem

twitter.com/NextFEM

linkedin.com/company/nextfem

NextFEM SRLS

Piazza del Foro Romano 12, 31046 Oderzo (TV) Italy - P.IVA 04954290260 - REA TV-413297