

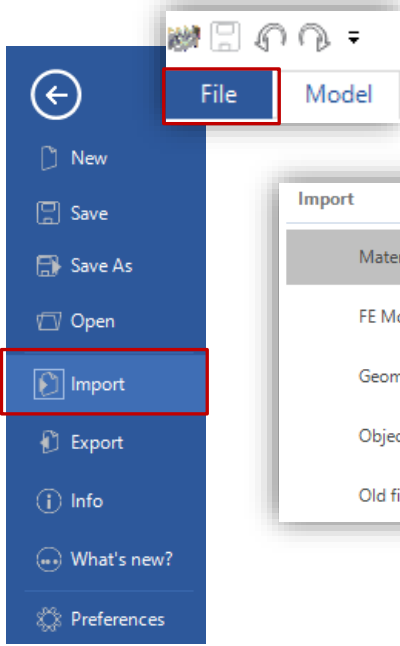
# manual import & export

  
impact  
design  
europe

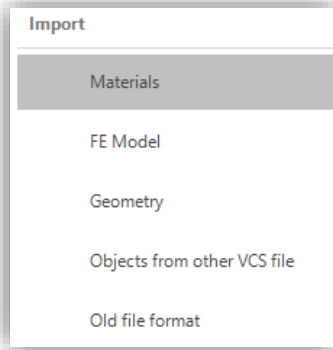


*The fastest way in Pre Design*

➤ <b><u>VCS IMPORT PROCEDURES</u></b>	<b>3</b>
▪ <u>Import Materials</u>	3
▪ <u>Import FE Models</u>	4
▪ <u>Import Geometry - STL Model</u>	8
▪ <u>Import Geometry – C-S from Hyperbeam</u>	9
▪ <u>Old File Format</u>	10
▪ <u>Import Objects from another VCS Solution</u>	11
➤ <b><u>VCS EXPORT PROCEDURES</u></b>	<b>13</b>
▪ <u>Export STL Model</u>	13
▪ <u>Export Nastran HyperWorks</u>	14
▪ <u>Export Nodes Position To Text</u>	14
▪ <u>Selected Objects to new VCS file</u>	15



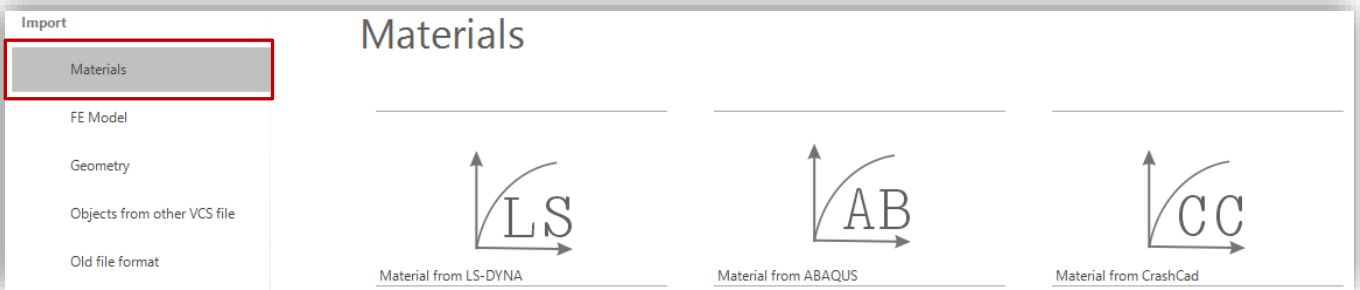
In order to enter VCS import menu, click on **File** and then on **Import**. From the drop-down list select the required import option.



Basic import options available in VCS:

- Material import
- FE model
- Geometry
- Objects from another VCS file
- Old file format

## IMPORT MATERIALS

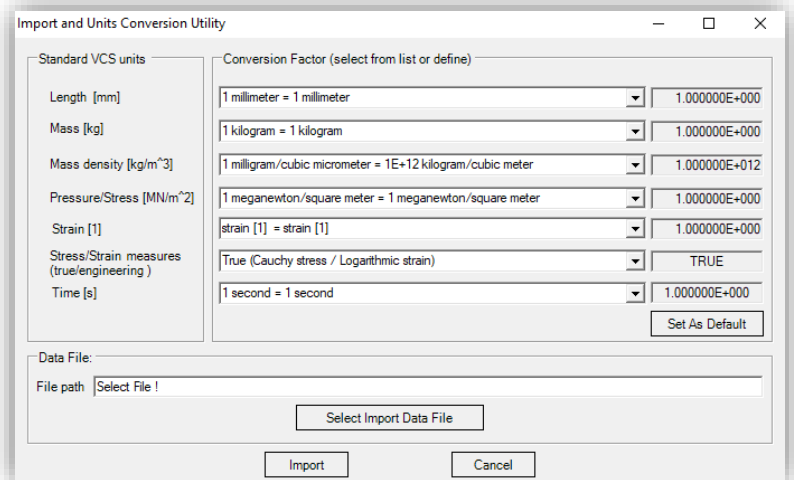


VCS provides the possibility to import material directly from LS-DYNA, ABAQUS or Crash Cad.

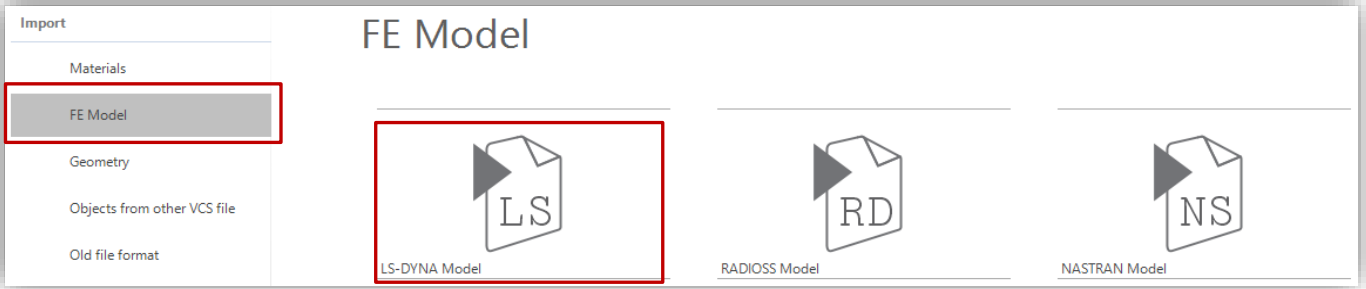
In order to import a material, click on **File** and then on **Import**. Select Material and afterwards the proper option (Material from... LS-DYNA, ABAQUS or Crash Cad), as it is illustrated in the picture above.

In "Import and Units Conversion Utility" window set proper units for the material properties. Once the units are set, select a file you wish to import. After that click on the **Import** button.

Imported Materials will appear in the Solution Explorer tree.



## IMPORT FE MODEL



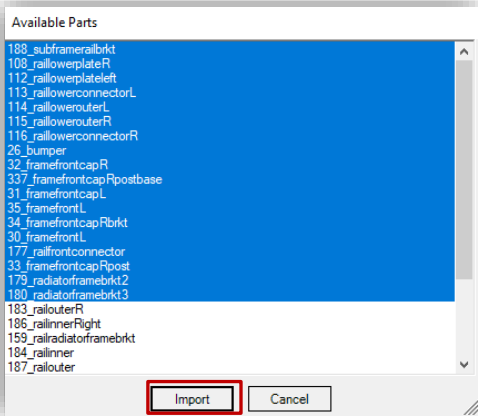
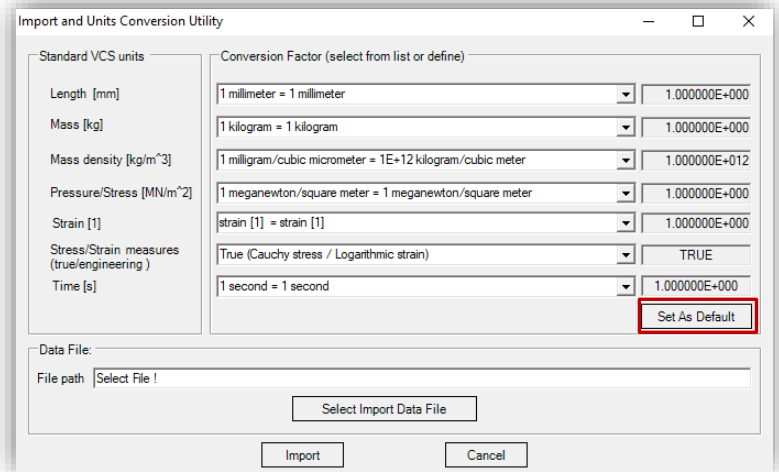
VCS gives the possibility to import data from a FE models such as LS-Dyna, Radioss, Nastran. The **"FE Model"** import procedure will be described on an example of a mesh model originating from the LS-Dyna software

VCS provides the possibility to set the required units when importing a FE model.

After selecting the proper import option, a new window **"Import and Units Conversion Utility"** will appear on the screen.

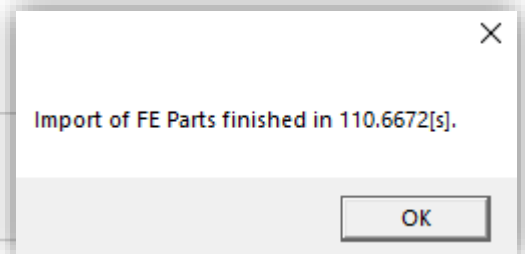
In this window you can set the required units of the imported model (length, mass, mass density, pressure / stress, strain).

You can save your setting as default settings by pressing the "set as default" button in the Import and Units Conversion Utility window.



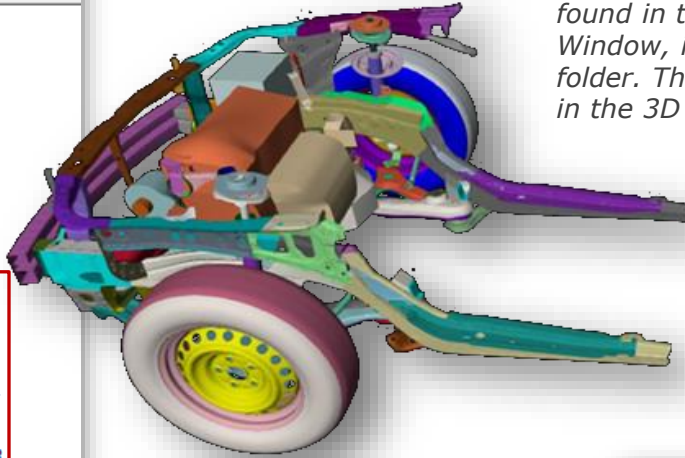
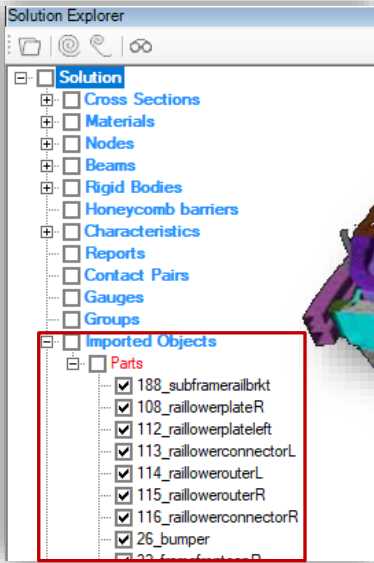
Once the units and scaling are defined you can select the Import Data file.

In the "Available Parts" window select all elements of the mesh model that are to be imported into VCS. Confirm by clicking on "Import"



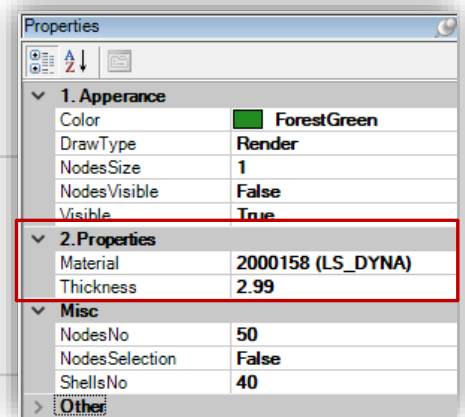
When import is finished the message box will appear on the screen. Click OK. The Import procedure is now finished.

## IMPORT FE MODEL



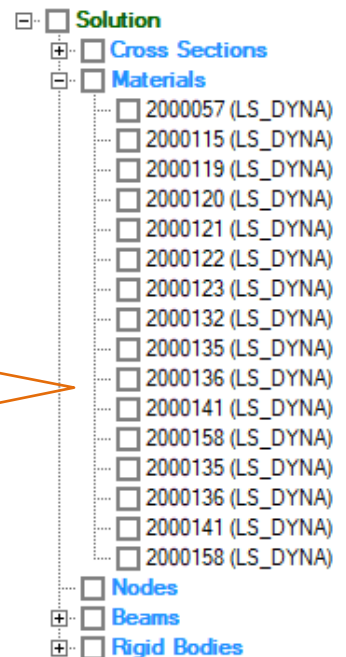
All imported elements can be easily found in the Solution Explorer Window, in the Imported Objects folder. They will be also displayed in the 3D view window.

Note that along with the geometry, all information about the Material of the FE elements as well shell thickness data will be automatically imported into VCS. They can be viewed in FE Part's Properties window (as shown in the picture on the right)

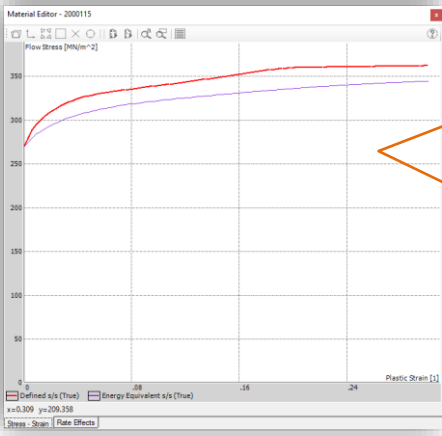


Moreover, the import tool creates simultaneously VCS Material basing on FE data. The Material Properties such as Material Constants, Stress - Strain -Characteristics and the Strain -Rate Effects parameters are visible in material definition.

All imported or created Materials can be easily found in the **Solution Explorer Window**



## IMPORT FE MODEL



All Materials used in the VCS Solution can be defined by means of the **Material Editor** window. The Material is described by five groups of data in the Properties window:

- Material Constants
- Stress Strain Characteristic
- Strain Rate Characteristic
- Hardening Factor
- Fracture Indicator

Material Properties such as :

- Material Constants,
- Stress – Strain - Characteristics and
- Strain – Rate Effects

parameters are visible in material definition in the Properties window.

<b>1. Material - Attributes</b>	
Type	DuctileIsotropic
<b>2. Material Constants</b>	
HardeningFactor	1
MassDensity	7890
PoissonRatio	0.3
ProofStrain	0.002
ProofStress	350
YoungModulus	200000
<b>3. Stress - Strain Characteristic</b>	
StressStrainCharacteristic	Array: [11] points defined
StressStrainCharacteristicType	Array
StressStrainHardeningLaw	100 [%] isotropic
StressStrainHardeningType	Isotropic
StressStrainMeasure	True
<b>4. Strain - Rate Effects</b>	
StrainRateCharacteristic	D=8 E+3 , p=8
StrainRateType	CowperSymonds
<b>5. Fracture Indicator</b>	
AreaReduction	0
D	1
FractureModel	SurfaceStrains
<b>Other</b>	
Comment	
Layer	Default
Name	2000187 (LS_DYNA)

### IMPORTANT NOTICE:

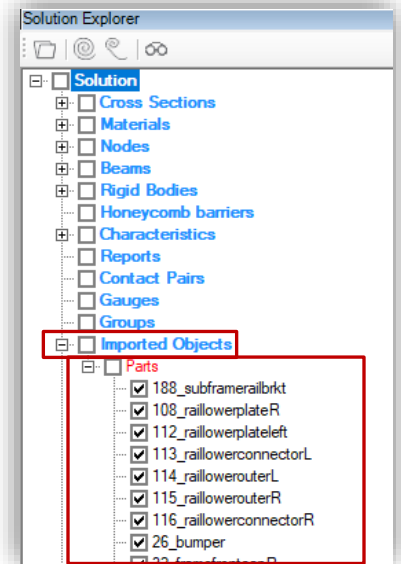
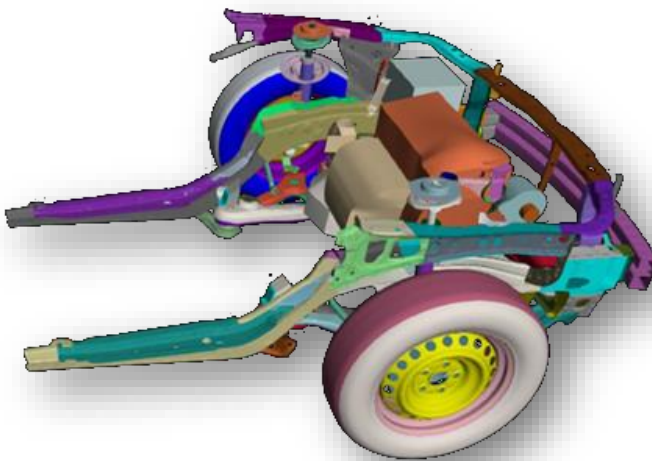
In case of material import from available FE models material properties must be verified. Improperly defined Material is a common mistake that causes distortion of the simulation results!

## HANDLING IMPORTED FE MODEL

A VCS model can be created basing on an original FE geometry which can be easily imported from a mesh – based software. Among several import options, VCS offers the possibility to import geometry from Ls-Dyna which is used in the presented example.

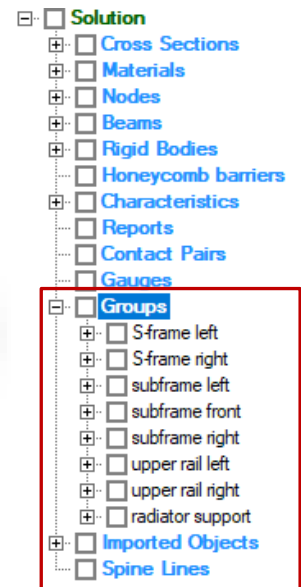
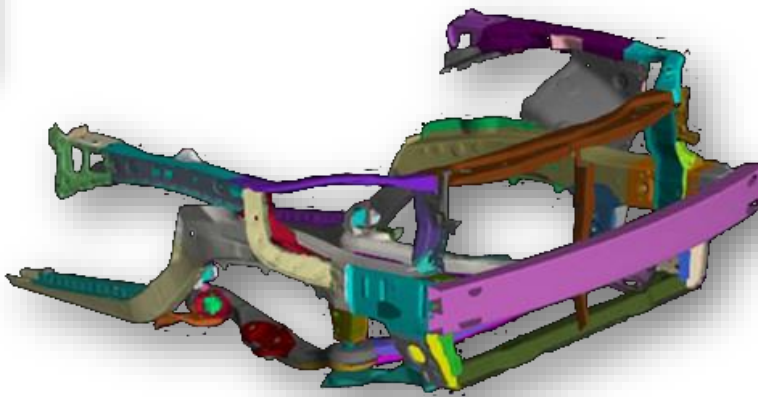
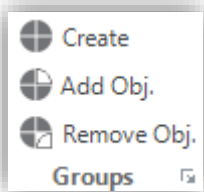
Note that along with the geometry, all information about the Material of the FE elements as well shell thickness data will be automatically imported into VCS. Moreover, the import tool creates simultaneously VCS Material basing on FE data.

All imported elements can be easily found in the Solution Explorer Window, in the **Imported Objects folder**.



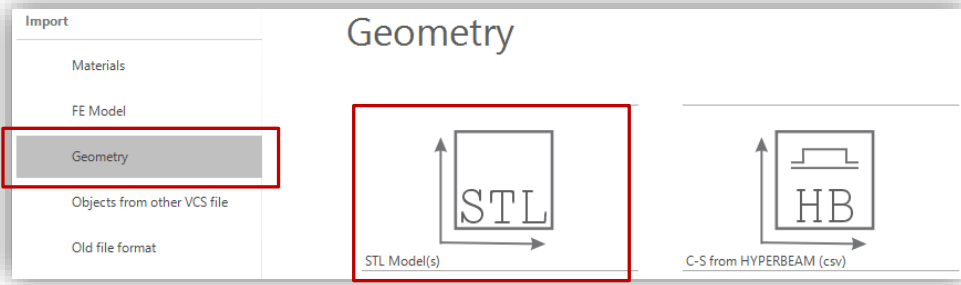
The first step recommended to be taken after FE geometry is imported into a VCS Solution is creating **groups of FE parts**.

Placing specific assemblies (for example bumper, s-frame, A pillar etc.) into separate groups is very useful during the later work and beam modelling.





## IMPORT GEOMETRY – STL MODEL



VCS offers the possibility to import a STL file format model.

In order to start the import procedure, click on File and then on Import. From a list select "Geometry" and afterwards the **STL Model** option.

A new **Import and Units Conversion Utility** will appear on the screen. In this window the user can set the required units of the imported model (length, mass, mass density, pressure / stress, strain).

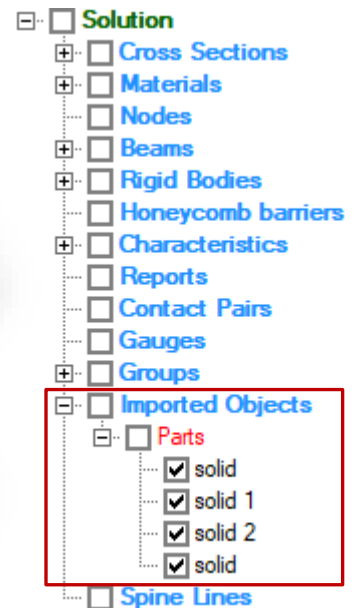
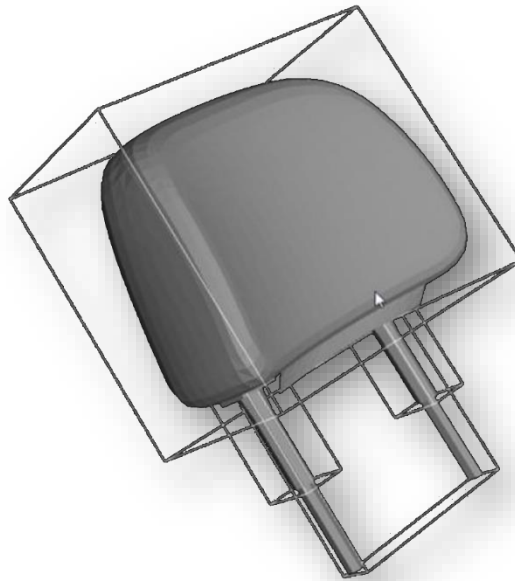
Note that you can save your setting as default settings by pressing the "set as default" button in the Import and Units Conversion Utility window.

In VCS the user has the possibility of multiple file import which means that number of elements can be selected during one import procedure.

Select as many STL elements as required and click on the **Open** button.

All imported elements will be automatically saved in the Solution Explorer tree under the Imported Objects ⇒ Parts folder.

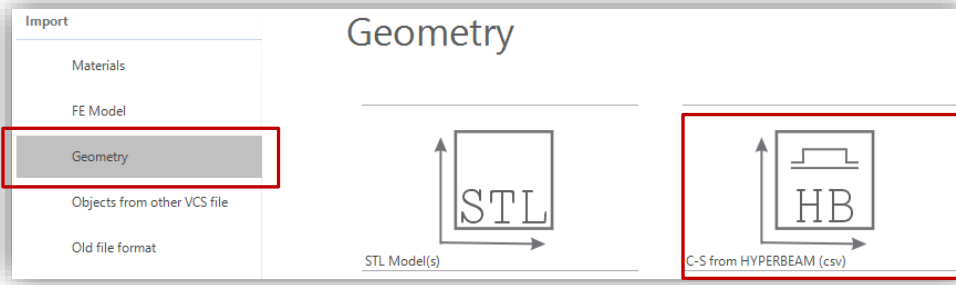
They will be also visible in the Solution 3D view.



Please note that STL geometry import does not include information about used materials or elements thickness.



## IMPORT GEOMETRY – C-S FROM HYPERBEAM

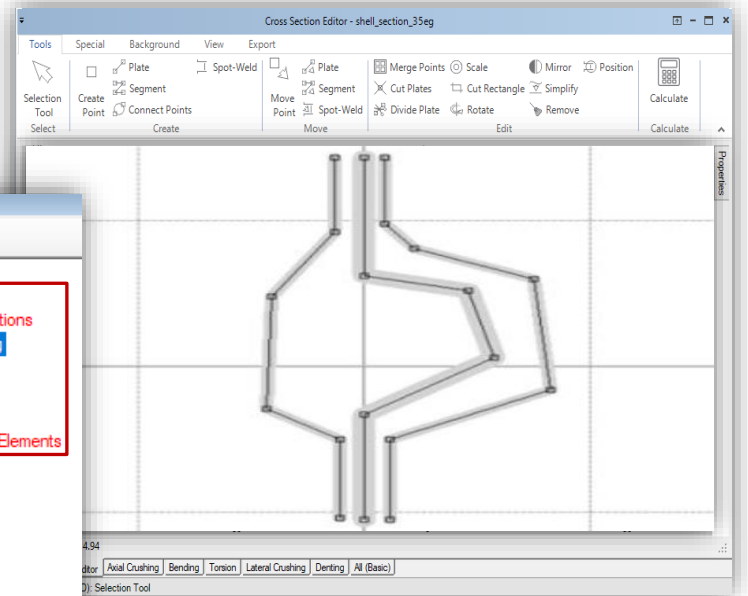
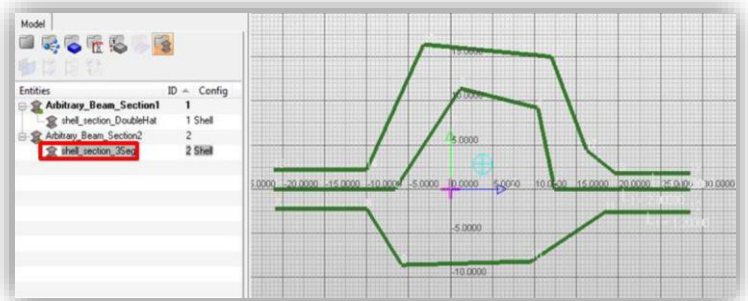


VCS offers the functionality of import from the HYPERBEAM software. You can import a cross section which was prior saved in HYPERBEAM in CSV file format.

After opening the VCS "File" menu select "Import" and then the "geometry" section. Afterwards select the "C-S from HYPERBEAM csv" option

After selecting the required "csv" file, all included cross sections will be imported into VCS.

The Cross Sections are going to be added to the Solution Explorer tree. In the pictures on the right, you can see an example of a cross section created in HYPERBEAM, and its counterpart automatically created in VCS as a result of the import procedure



The geometry has been automatically created and can be viewed in the Cross Section Editor. All elements are also automatically added to the Solution Explorer tree. Please note that the imported cross section is already divided into segments, plates and points.



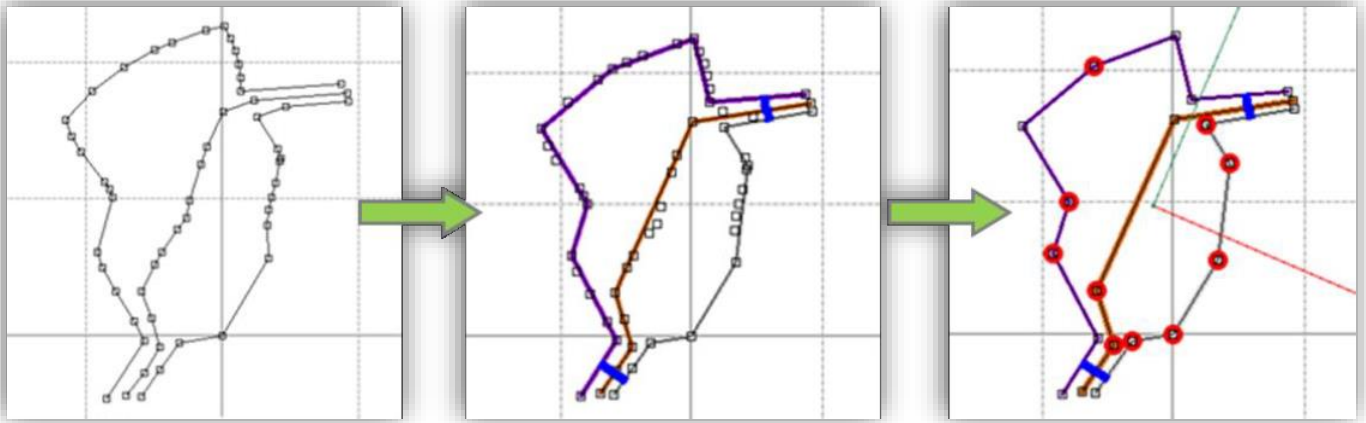
## IMPORT GEOMETRY - C-S FROM HYPERBEAM

The newly imported, crude cross-sectional geometry needs to be modified to fulfill requirements of macro element methodology!

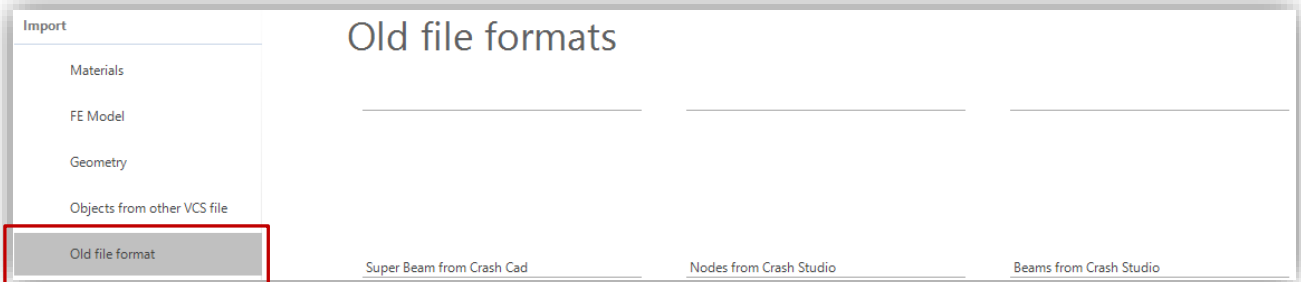
Use Cross Section Editor's tools to simplify the original geometry:

- Modify and simplify the original cross-sectional geometry – you can use the "Move Point" tool
- Add Connections between segments
- Remove not connected Points

It is important to underline that before cross section calculations can be conducted, material needs to be assigned to each segment



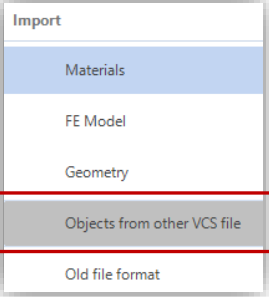
## OLD FILE FORMAT



VCS enables import of objects created and saved in old versions of the Crash Cad and Crash Studio software:

- Import Super Beam from Crash Cad
- Import Nodes from Crash Studio
- Import Beams from Crash Studio

## IMPORT OBJECTS FROM OTHER VCS SOLUTION



In order to import objects from a previously created VCS model go to "File" menu and click on "Import". Select the "Objects from other VCS file" option.

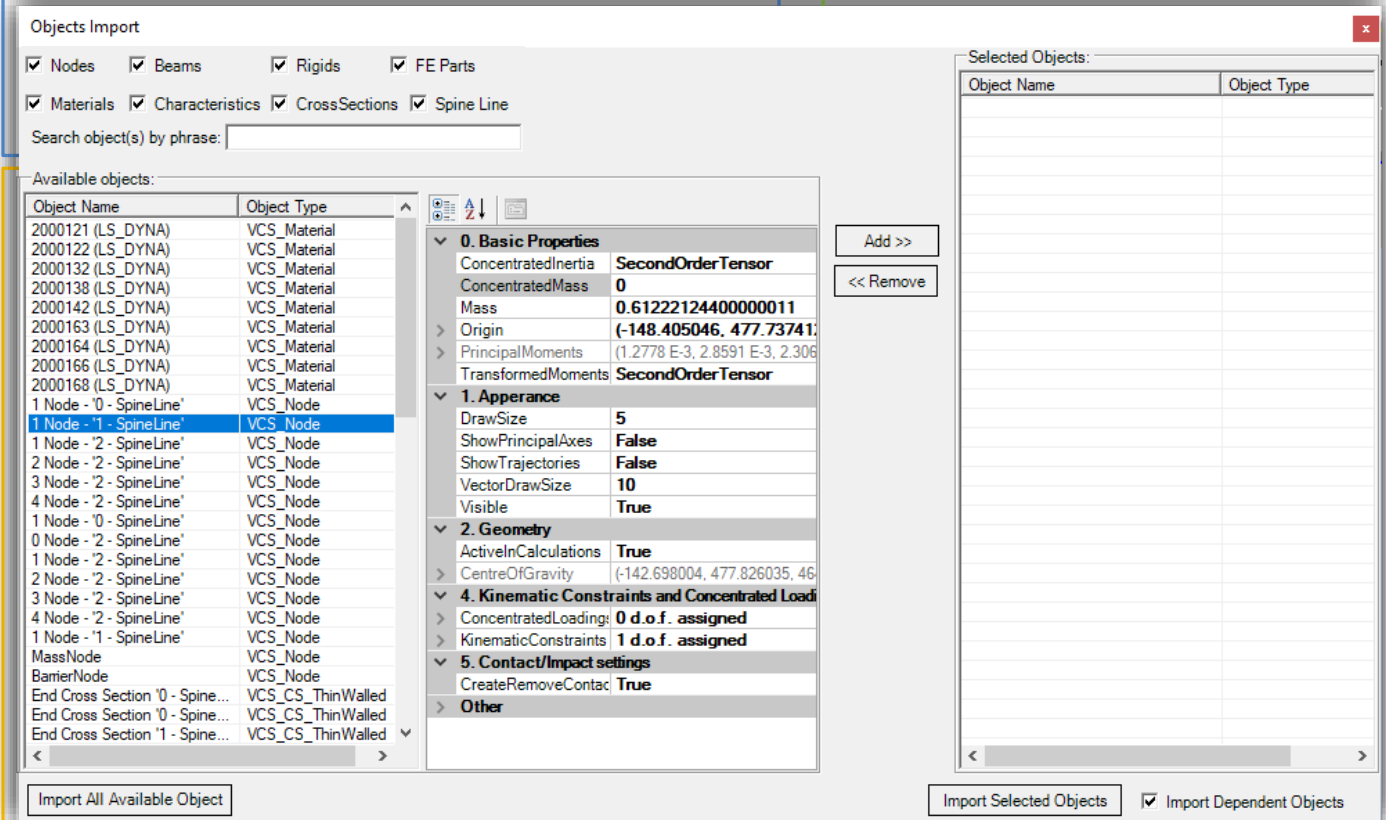
Select a VCS Solution from which you wish to import objects and press **Open**.

An **Objects Import** window will appear on the screen. It contains a list of all objects created in the opened Solution.

You can indicate type of objects to be displayed on the list by clicking specific check-boxes. Only displayed objects will be imported to the VCS.

You can search objects by phrase. After entering a phrase in the textbox, only the objects that contain it, appear in the list. Please note that search is case sensitive.

You can import only selected objects as you wish. After selecting them from the 'Available objects' list, click on the **Add** button to add them to Selected Objects list. To delete unnecessarily added objects select them from the Selected Objects list and click on the **Remove** button.



On the left side of the window, you can find the list of all available objects to be imported. On the right side, you can find a read only Properties window showing detailed information about a selected object.

You can import the entire Solution at once by clicking on the **Import All Available Objects** button.

When selection is finished click on the **Import Selected Objects** button.

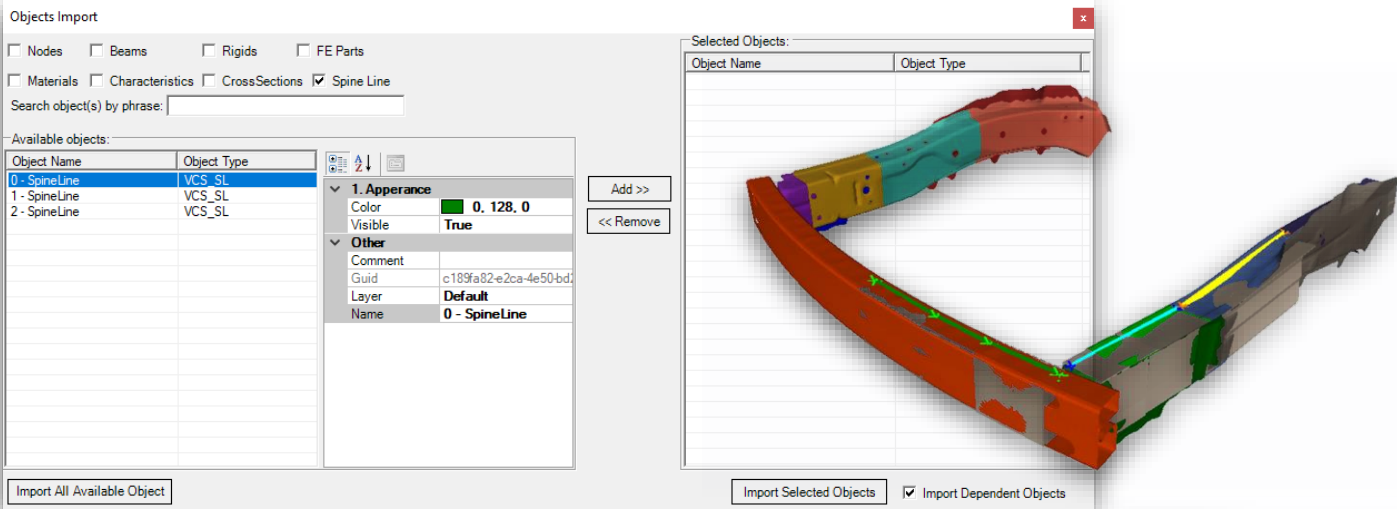
To import objects related with selected ones (e.g., Cross Sections assigned to selected Super Beams) choose **Import Dependent Objects**.

The imported objects will be automatically added to the Solution Explorer tree. They will also appear in the 3D view window.

## IMPORT OBJECTS FROM OTHER VCS SOLUTION

### Spine Line import

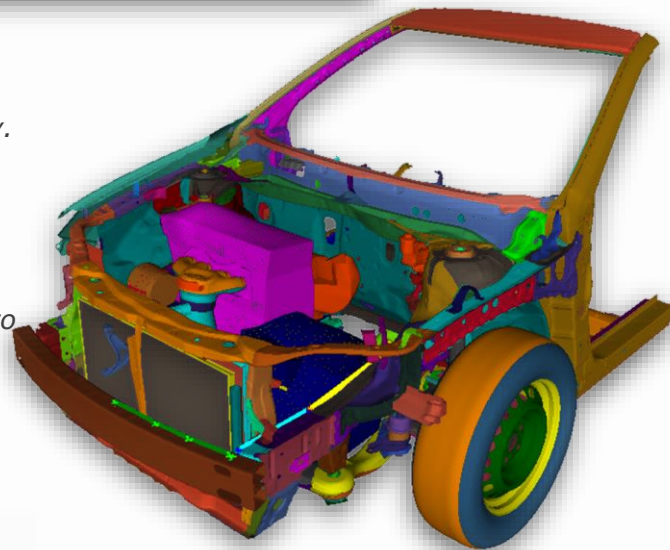
The Spine Line functionality enables easy usage of previously prepared discretization for the needs of another solution.  
Spine Line can be imported from another VCS solution.



If you have a solution with an already discretized model, similar to the currently used one, or a fragment (subassembly) of this model, you can easily use it with the help of the import functionality. This saves time for planning the discretization.

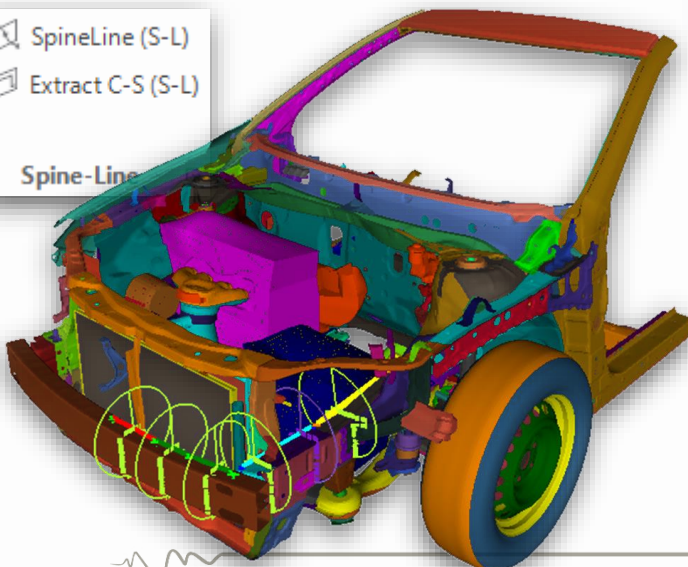
It is possible to import selected Spine Lines. Automatically, the Points positions are imported, all Lines contained in a SL, as well as the position of cutting planes. Additionally, all information related to the appearance of the imported objects set in the reference solution is captured.

Remember that the imported Spine Lines may require some modifications before creating Super Beams on its basis.



- SpineLine (S-L)
- Extract C-S (S-L)

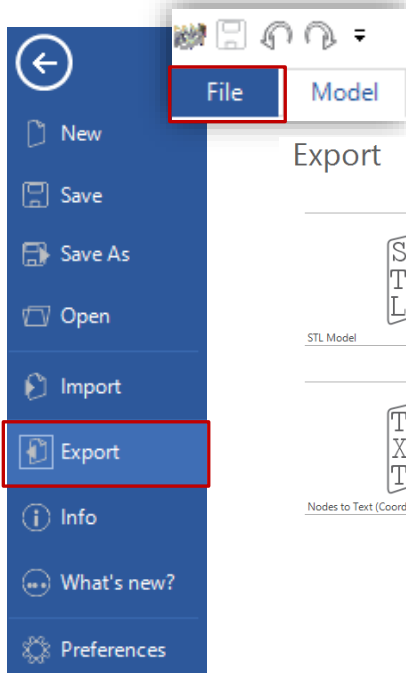
Spine-Line



Repositioning the SL Points is easily accomplished by dragging and dropping them with the Spin Line tool (see the Spine Line tool - 3D Modeling Manual).

Note that all visible objects inside the Circle will be included in the future Super-Beam's section.

The radius and position of the Circle as well as visibility of FE Parts should be adjusted to include all elements relevant for the future cross section. (See the Spine Line tool - 3D Modeling Manual).



In order to enter VCS Export menu, click on **File** and then on **Export**. From the drop-down list select the required export option.

## Export



STL Model



Nastran HyperWorks



Selected objects to New VCS file



Nodes to Text (Coordinates)

### Basic export options available in VCS:

- STL Model
- Nastran HyperWorks
- Selected Objects to new VCS file
- Nodes to Text (Coordinates)

## EXPORT STL MODEL

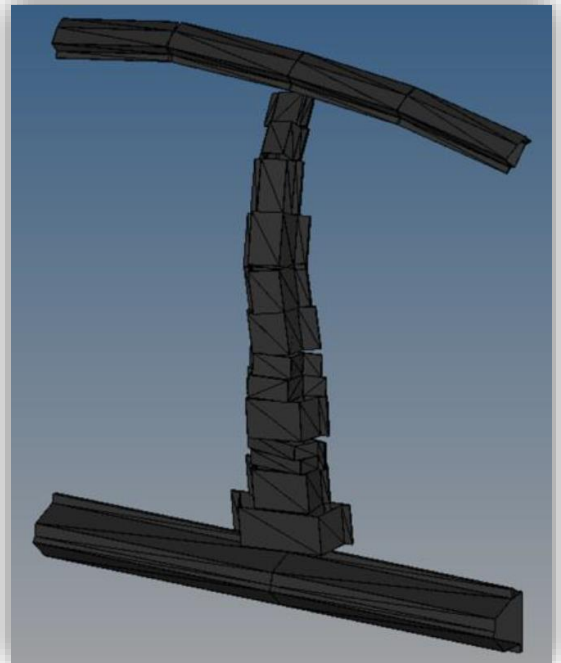
Any structure created in VCS 3D environment can be easily exported as a STL model.

In order to export the 3D geometry, click on **File** and then on **Export**. Choose the **STL Model** option.

You can save the exported STL model in any location and name it as you wish.

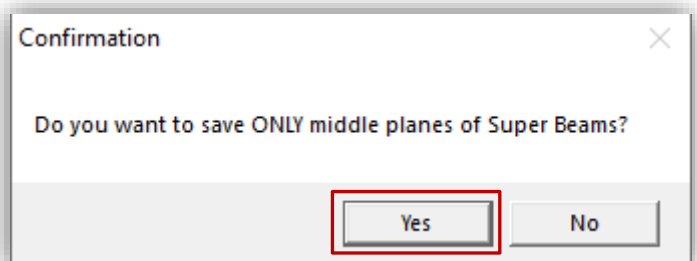
The result of the export procedure can be seen in the picture on the right.

Please note that STL geometry model does not include information about used materials or elements thickness.



The Export STL option enable to save original 3D geometry or only its middle planes.

To save only the middle planes of the geometry, choose YES button from the message box presented on the right.



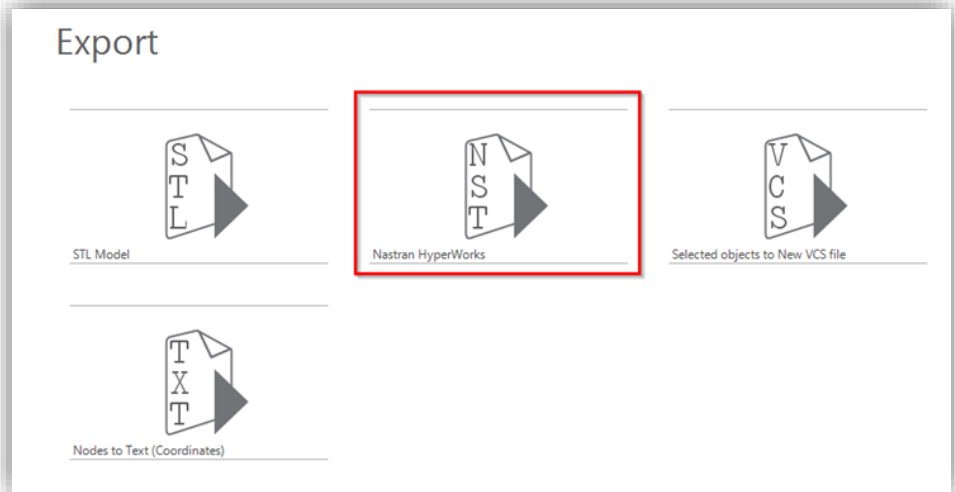


## EXPORT NASTRAN HYPERWORKS

Any structure created in VCS 3D environment can be easily exported as a Nastran model.

In order to export the 3D geometry, click on **File** and then on **Export**. Choose the **Nastran HyperWorks** option.

You can save the exported model in any location and name it as you wish.



Please note that export to Nastran format affects the data structure:

- only Super Beam Elements with Thin-Walled Cross Sections are exported
- only Cross Sections with uniform material are exported
- only VCS materials with stress-strain curve defined as Array2D are exported

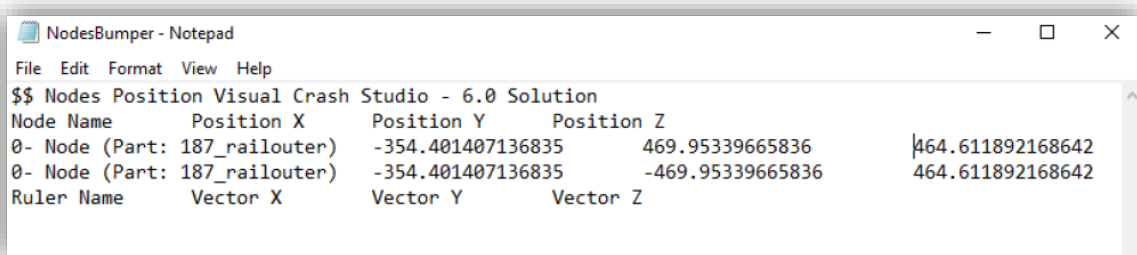
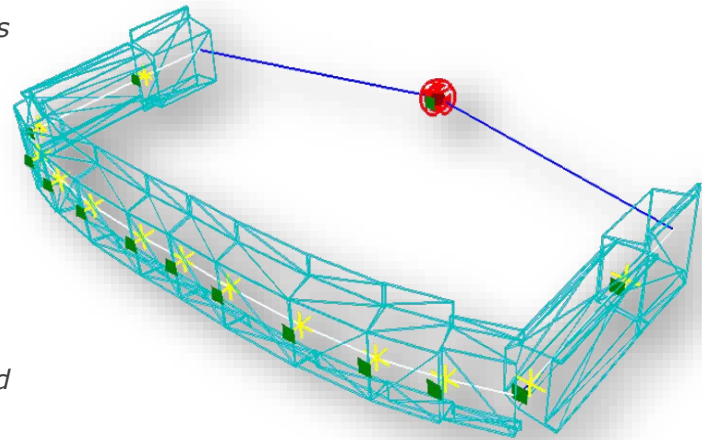
## EXPORT NODES POSITION TO TEXT

VCS enables the user to easily export coordinates of selected nodes to text files.

In order to start the export procedure, select the original VCS model all Nodes which coordinates are to be exported.

Afterwards go to **File** and then to **Export**. In the export menu select **Nodes to Text (Coordinates)** option.

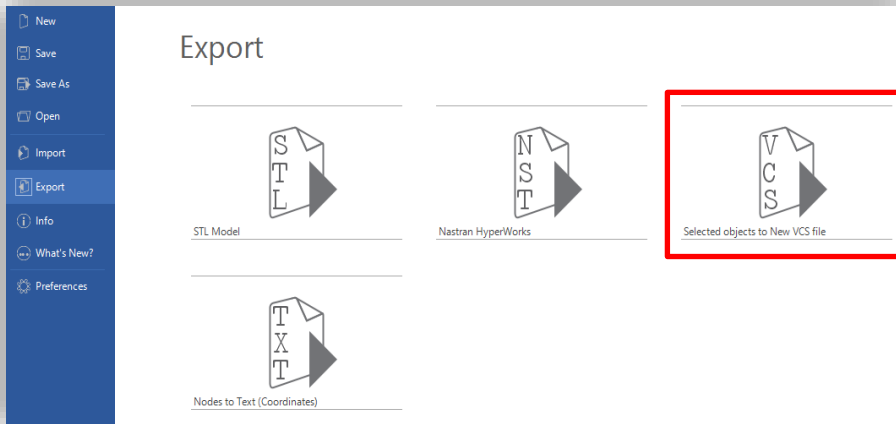
Coordinates of all selected Nodes will be exported po a text file format (as illustrated in the picture below)



## EXPORT SELECTED OBJECTS TO NEW VCS FILE

Any object created in VCS 3D environment can be easily exported to the new solution.

If there is a need to focus on some specific parts or subassemblies of the previously created complex structure, in an easy way they can be exported to a new solution and analyzed independently.



To export VCS objects to the new solution, select objects to be exported and go to **File -> Export -> Save Selected VCS Objects**.

Objects can be selected in the 3D window or in the Solution Explorer tree.

The new solution with exported objects can be save in any location chosen by the user.

The export procedure is presented below on the example of extracting bumper subassembly from complex front structure.

1. In order to analyze bumper subassembly separately, select bumper beams and every object to be exported
2. Chose **Save Selected VCS Objects** option
3. Show the destination of New Solution
4. Add the name of New Solution
5. Selected objects can be found in the newly exported VCS file.

