manual 8
3D simulations

impact design europe

The fastest way in Pre Design

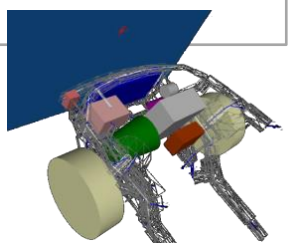
- ***VCS 3D SIMULATIONS – WORKFLOW*** **3**
- ***MASS DISTRIBUTION*** **4**
- ***BOUNDARY CONDITIONS & INITIAL CONDITIONS*** **5**
  - *Kinematic Constraints* 6
  - *Setting Boundary Conditions – Examples* 9
  - *Concentrated Loadings* 10
  - *Assign Initial Velocity* 11
- ***CONTACT SETTINGS – BASIC INFORMATIONS*** **12**
  - *Contact Envelopes* 14
  - *Rigid Bodies as Contact Elements* 14
  - *Spherical Contact Interface* 15
  - *Contact Settings – Create Contact Bodies* 16
  - *Contact Box & Contact Cylinder* 16
  - *Contact Envelopes for Beam Elements* 17
  - *Contact Pairs definition – Step by Step* 18
  - *"Create Contact Pair" tool* 18
  - *Contact Setting in Elementary Model – Example* 22
  - *Contact Zones* 23
  - *Contact Pairs Settings – Examples* 24
  - *General Contact* 26
  - *General Contact – Settings Examples* 27
- ***"CREATE DEFORMABLE BARRIER" TOOL*** **29**
- ***SOLUTION SETTINGS*** **31**
  - *Time Stepping Routine – Step by Step* 33
- ***SIMULATION & ANIMATION*** **34**
  - *Calculations – Multithread* 34
  - *Animation* 35
  - *Animation Toolbar* 36
  - *Synchronization Functionality* 37
  - *Chart Wizard* 39

VCS 3D SIMULATIONS – WORKFLOW

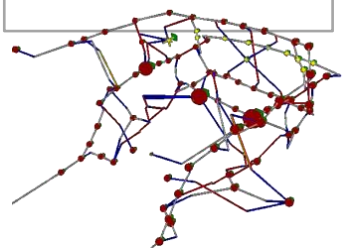
The specific of the VCS software requires distributing the mass of a structure into Nodes. Additional mass can be added by definition of concentrated mass in node properties or creation of rigid bodies with defined mass/or density. Boundary conditions settings in VCS are available in Node's properties window (Kinematic Constraints and Concentrated Loadings section).

The contact/impact can be simulated for all structural VCS elements and all their combinations. After calculations are done, the Chart Wizard is used to create various graphs illustrating the simulation results.

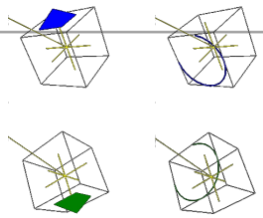
Additional elements



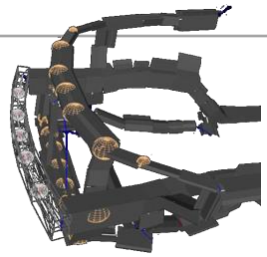
Mass Distribution



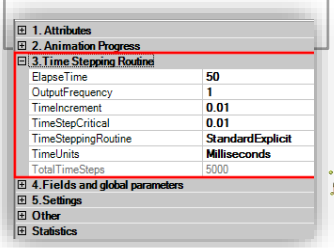
Initial & Boundary conditions



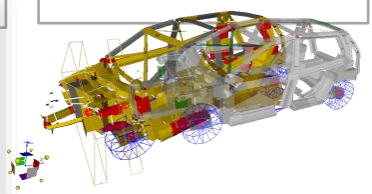
Contact Settings



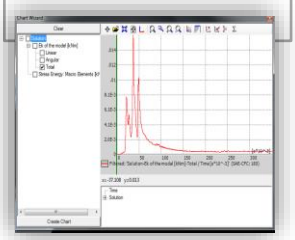
Solution Settings



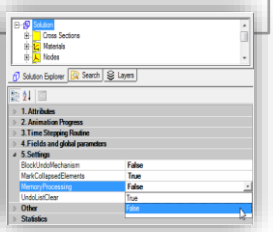
Calculations & Animation



Results – Chart Wizard



Export

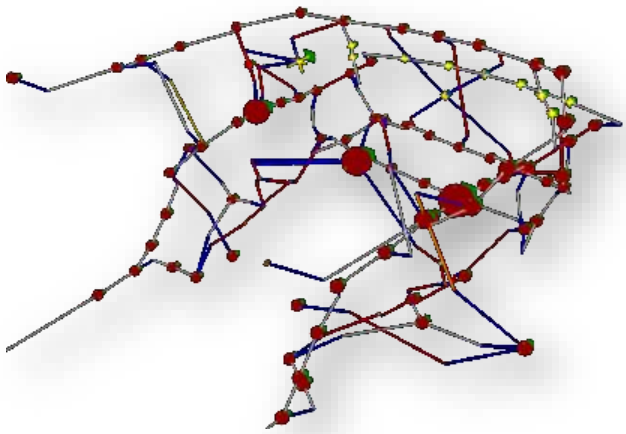


Results of a VCS Simulation are available in Chart Wizard, which is described in the "VCS Chart Wizard Manual"

For more information about export possibilities available in VCS please see the "VCS Import & Export Manual"

MASS DISTRIBUTION

*The specific of the VCS software requires distributing the mass of a structure into **Nodes**.*

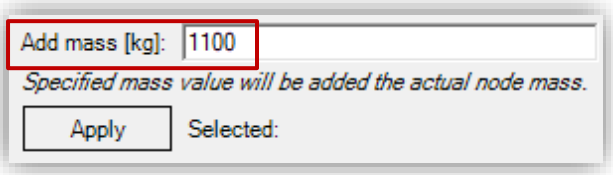


The information about the mass of each object created in the 3D environment is located in the Node/s on which the given element was defined.
In consequence modeled beam of total mass of 2 kg assigns automatically a 1 kg mass to each of the Nodes on which the beam is created.

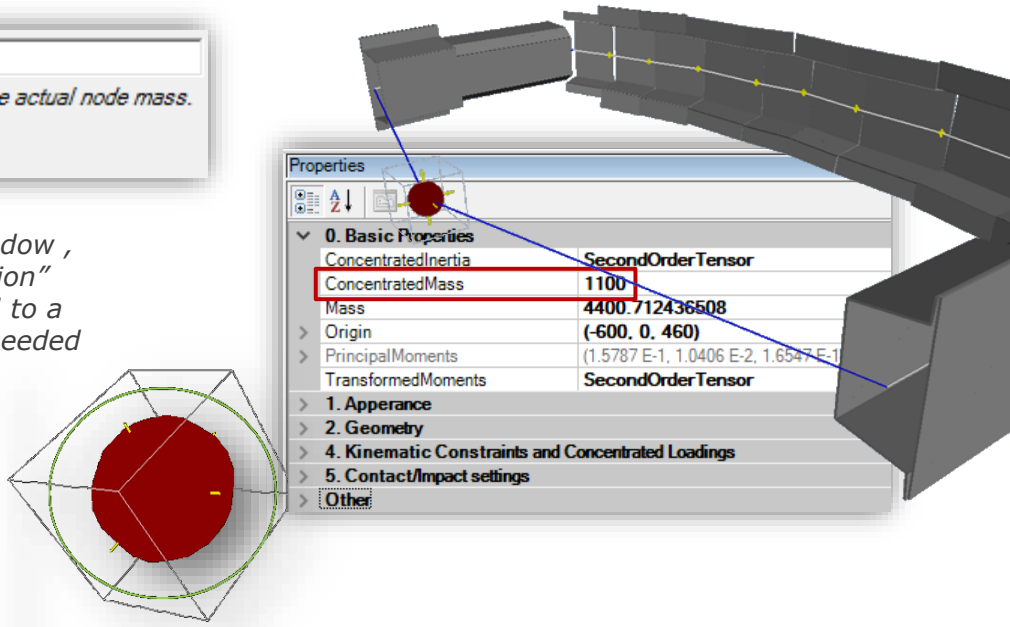
Additional mass can be added by definition of concentrated mass on a selected Node.

Additional concentrated mass can be defined for a Node by means of the "Add Concentrated Mass" tool in following steps:

1) Select the "Add Mass" tool from the main toolbar
2) In the 3D view select the Node to which you wish to add mass
3) Define mass value in the tool settings window, confirm by clicking "Apply"
4) Additional concentrated mass will be visible in the 3D view (red sphere). Additionally, it will be available in the Node's Properties window



In the Node's Properties window , in the "Basic Properties section" concentrated mass assigned to a Node can be viewed and if needed edited.



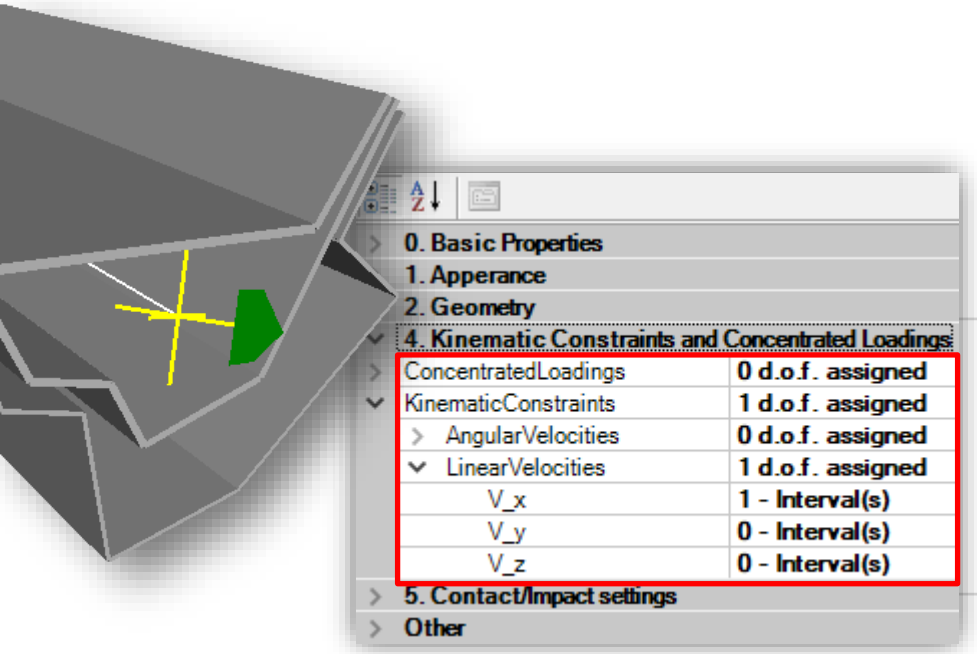
Mass and Inertia

Concentrated Inertia *→ The concentrated inertia tensor shows dialog window for user defined inertia (in the frame parallel to global frame of the model). This value is **added** to the inertia resulting from mass reduction of **all elements** attached to the node object. The inertia tensor is symmetric therefore only 6 entries are active in the editor of Second Order Tensor:*

Concentrated Mass *→ User defined concentrated Mass attached to the **Centre of Gravity** of the Node. (kg)*

Mass *→ total Mass of the Node*

BOUNDARY CONDITIONS & INITIAL CONDITIONS



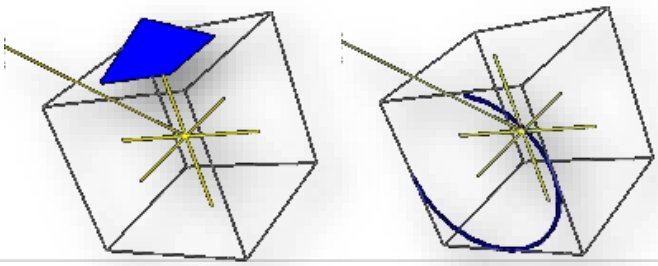
*Boundary conditions settings in VCS are available in Node's properties window (**Kinematic Constraints and Concentrated Loadings** section).*

The modeled structure may be exposed to specified loading which acts in time of simulation or may have limitations to the movement range.

Except the definition of forces, moments and velocities applied to certain Nodes of the modeled structure or reduction of degrees of freedom there is a possibility to simulate the vector of body forces. Definition of body forces vector is available in the Solution's Properties window (top most object in solution explorer tree).

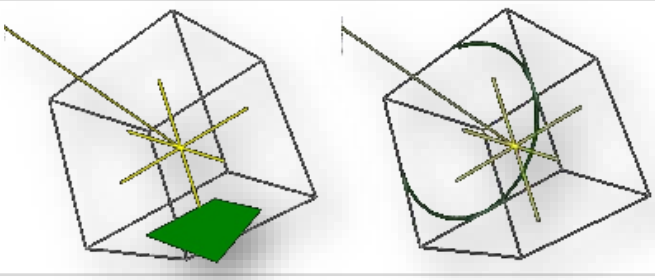
***Concentrated loadings** sections deals with forces and moments applied to selected Node or group of Nodes*

- *Forces*
- *Moments*



***Kinematic constraints** section is used to assign velocity history.*

- *Angular Velocities*
- *Linear Velocities*



KINEMATIC CONSTRAINTS – DETAILED INFORMATION

After choosing the 'Kinematic Constraints' field in the Kinematic Constraints and Concentrated Loadings section of the Node Properties window, you can set the same Kinematic Constraints (angular and linear velocities) to all 6 degrees of freedom.

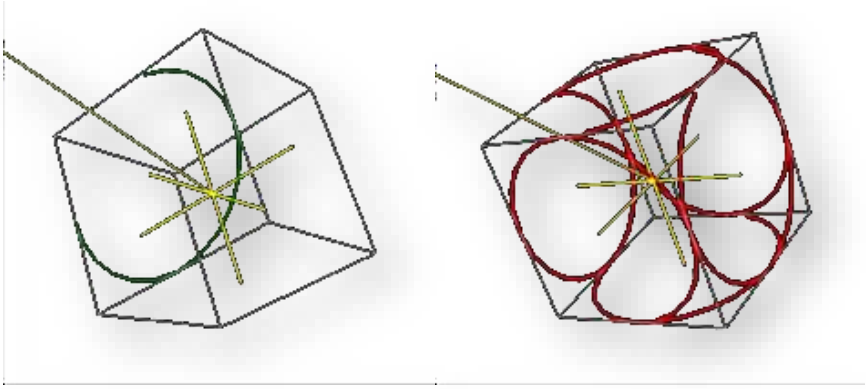
- *After choosing the **'Angular Velocities'** field you will set the same kinematic constraints to Angular Velocities for all d.o.f. or for a chosen degree of freedom*

After selecting the Omega X, Omega Y or Omega Z field of the Angular Velocities section of the Properties window you can define separate settings of angular velocities for X, Y and Z axis of the coordinate system.

After selecting the 'Fixed' option if the Kinematic Constraints window you can block Angular velocities for selected d.o.f

On the Solution 3D view the defined Angular Velocity will be illustrated by a round green line.

Fixed Angular Velocities will be illustrated by round red line (one line for each X, Y and Z d.o.f).



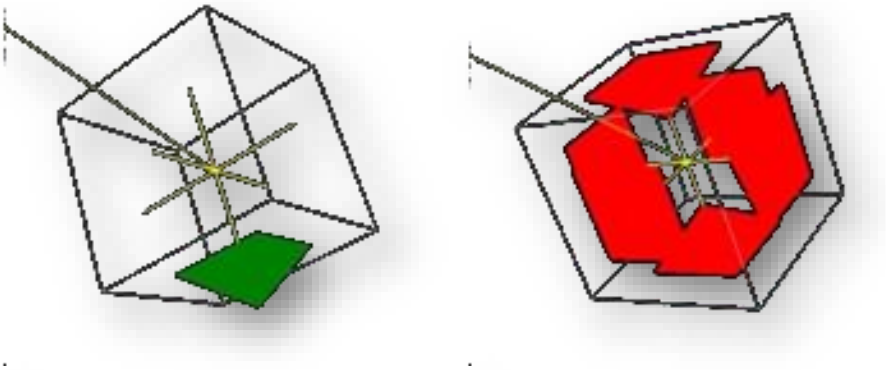
- *After choosing the **'Linear Velocities'** field you will set the same kinematic constraints to Linear Velocities for all d.o.f. or for a chosen degree of freedom.*

After selecting the V_x, V_y or V_z field of the Linear Velocities section of the Properties window you can define separate settings of linear velocities for X, Y and Z axis of the coordinate system.

After selecting the 'Fixed' option if the Kinematic Constraints window you can block linear velocities for selected d.o.f

On the Solution 3D view the defined Linear Velocity will be illustrated by a green arrow.

Fixed Angular Velocities will be illustrated by red arrows (one for each X, Y and Z d.o.f).

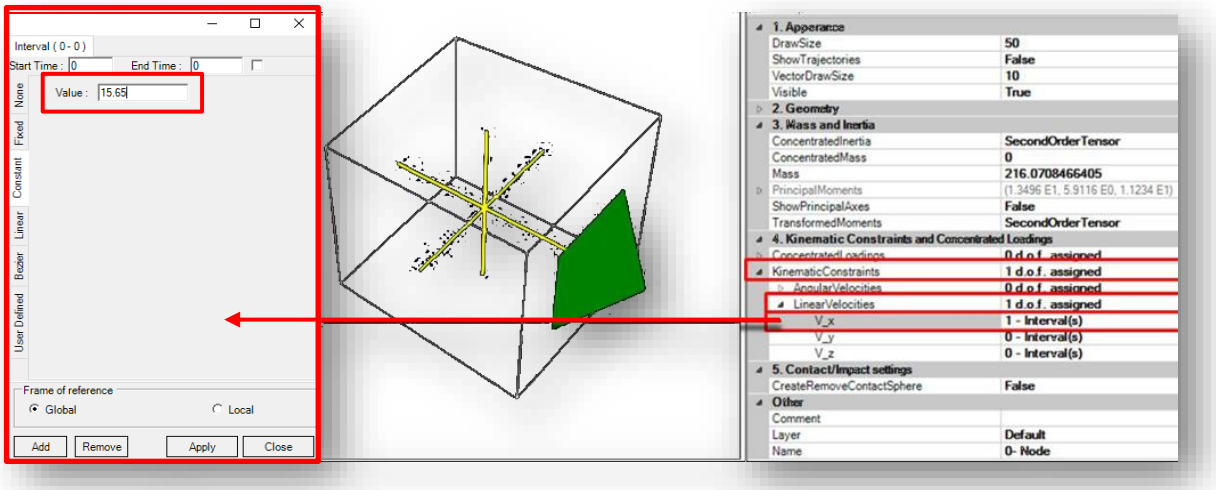


KINEMATIC CONSTRAINTS – DETAILED INFORMATION

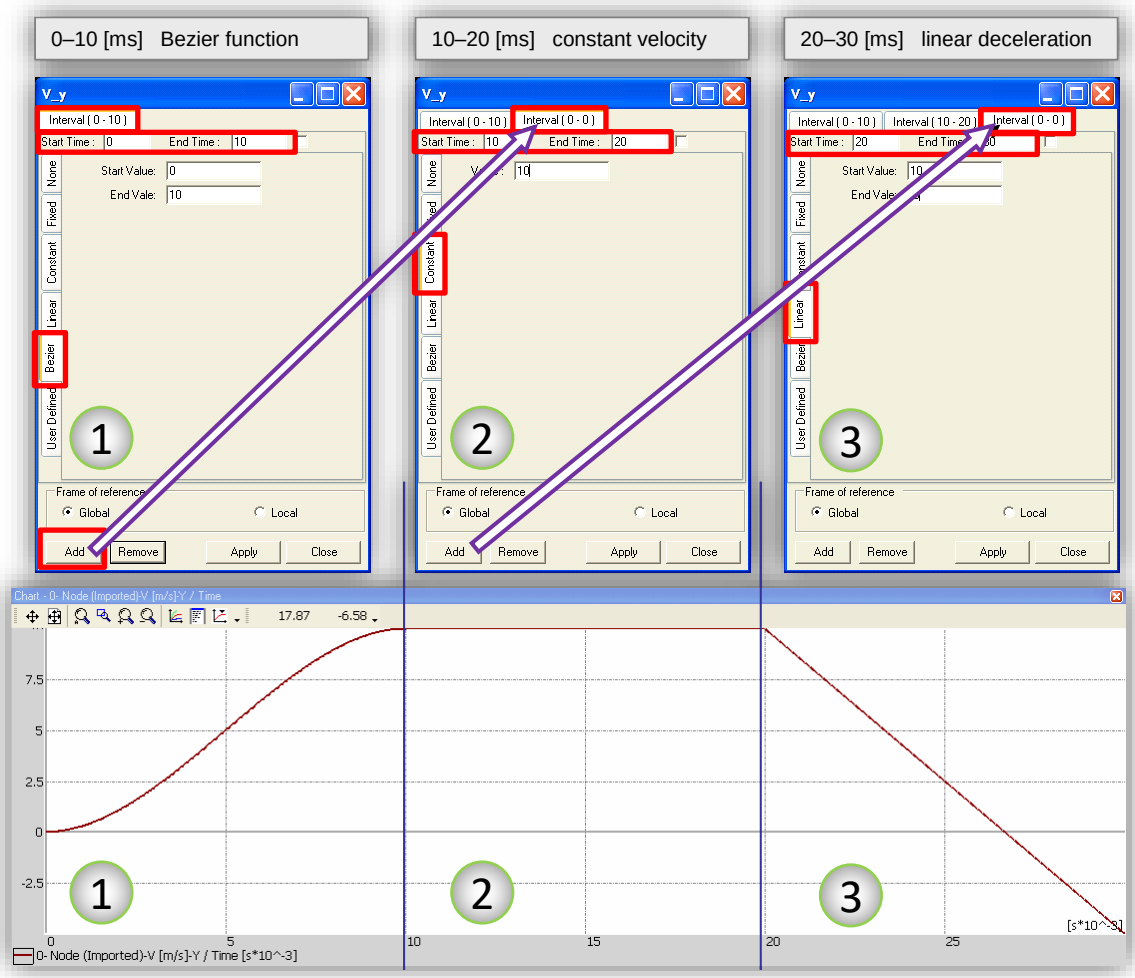
- *You can also set kinematic constraints only for a single d.o.f (for instance Y d.o.f in case of Linear Velocities)*

- *You can define parameters for selected history*
  For instance: constant positive velocity 12 [m/s]

- *You can define time interval for selected history.*
  For instance velocity during the entire simulation



- *You can also define a complex time history – a history defined separately in a number of time intervals – please see an example below*



KINEMATIC CONSTRAINTS – DETAILED INFORMATION

The procedure for definition of a predefined type of history is as follows:

1. *In the Kinematic Constraints and Concentrated Loadings section of the Properties window click on the Kinematic Constraints field. A window named Kinematic Constraints will be displayed on the screen.*

2. *Select frame of reference for the defined history:*

   - *global coordinate system*
   - *local Node principal coordinates (co-rational)*

   Click 'Add' to add time interval.

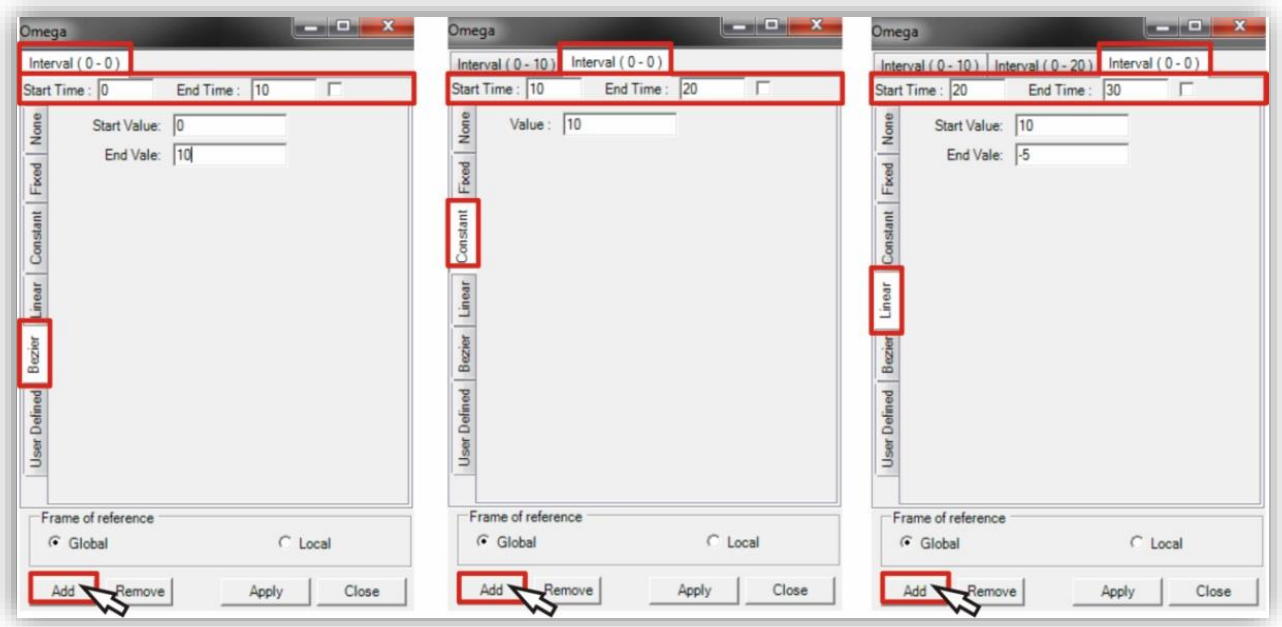
3. *Select type of time – history:*

   - *Fixed d.o.f. – specific for kinematic constraints corresponds to zero velocity*
   - *Constant velocity*
   - *Linear*
   - *Bezier – type history 2 order*
   - *User defined characteristic*

4. *Define parameters for selected history.*
   For instance: constant positive velocity 12 [m/s]

5. *Define time interval for selected history.*
   For instance velocity during the entire simulation (marc

In the example illustrated below kinematic loading starts from smooth, Bezier-type grow of velocity from 0 to 10 [m/s] during first 10 milliseconds followed by constant 10 [m/s] velocity during consecutive 10 millisecond and linear deceleration to -5[m/s] in the consecutive 10 milliseconds interval.



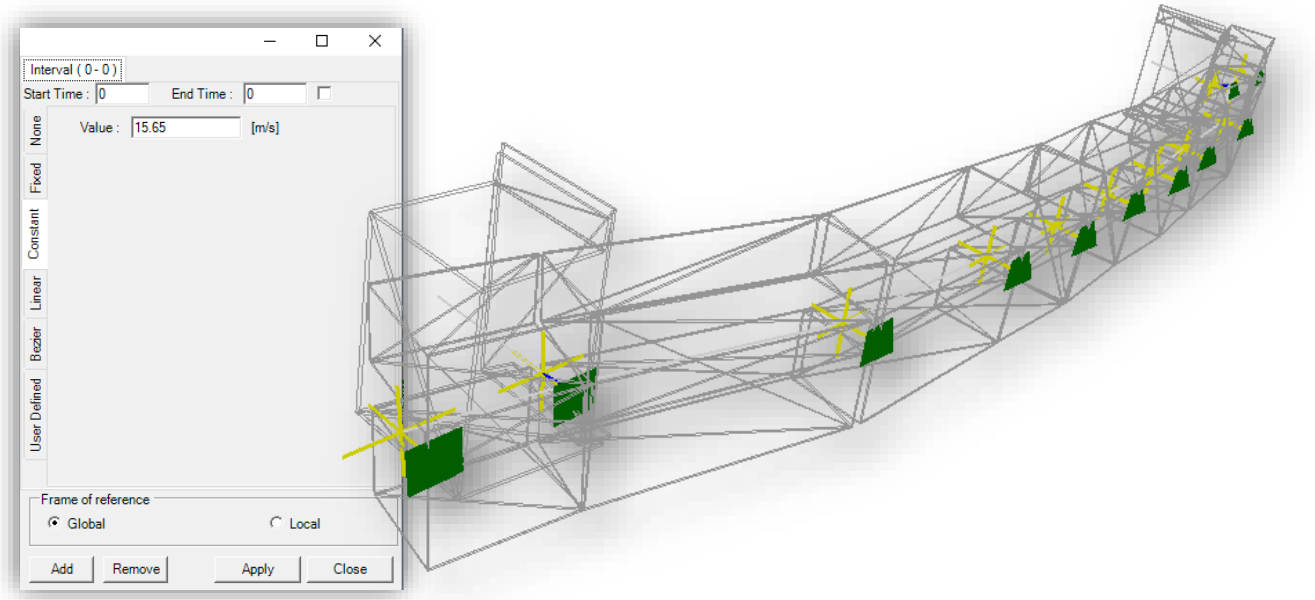
Following similar procedure you can set Kinematic constraints for chosen d.o.f.

INITIAL LINEAR VELOCITY – STRUCTURE NODES

Please find below an example of a simplified bumper structure with assigned linear velocity. This is a standard procedure while defining a VCS load case simulation.

In the presented case an initial velocity of 15.65 m/s has been assigned (velocity's start and end time has been set to zero).

The linear velocity is visualized as a green arrow visible on all nodes of the structure.

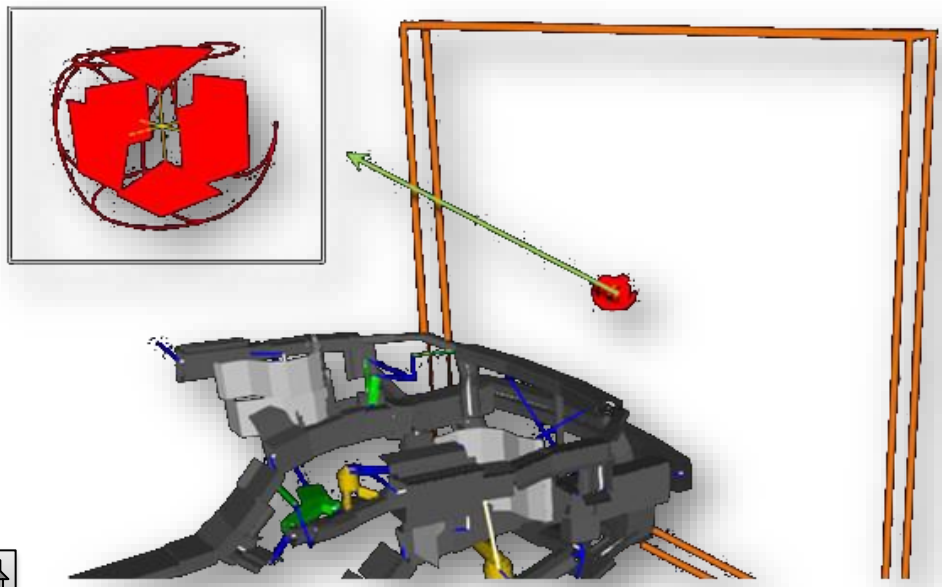


RIGID BODY BARRIER – EXAMPLE OF KINEMATIC CONSTRAINTS SETTINGS:

Special boundary conditions can be defined for the barrier Rigid Body Node.

As illustrated in the picture below, this Node was blocked in all degrees of freedom.

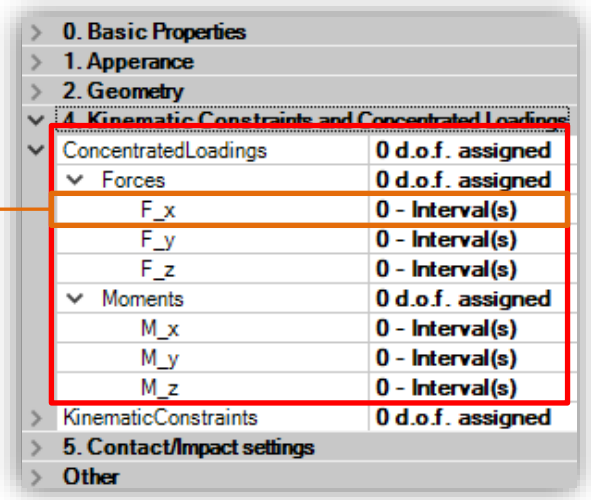
In other words, all angular and linear velocities were fixed in case of this Node.



CONCENTRATED LOADINGS

*In the **Concentrated Loadings** section of the Node's Properties window you can set forces and moments for all degrees of freedom or for a selected degree of freedom.*

The procedure of defining Concentrated Loadings is practically identical as the one described for Kinematic Constraints.



1. *After selecting one of the fields of the **Concentrated Loadings** section of the Properties window, a window named Loadings will be displayed on the screen.*

2. *Select frame of reference for the defined history:*

   - *global coordinate system*
   - *local Node principal coordinates (co-rational)*

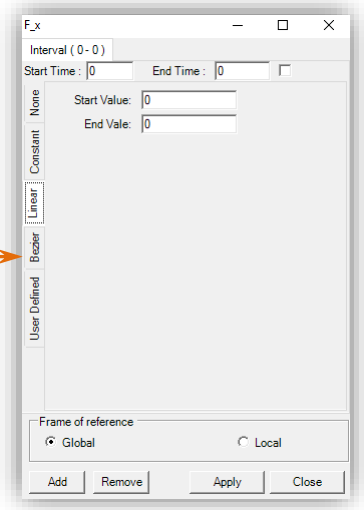
Click 'Add' to add time interval.

3. *Select type of loading*

   - *Constant*
   - *Linear*
   - *Bezier – type history 2 order*
   - *User defined characteristic*

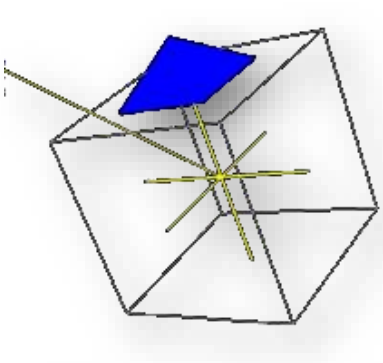
4. *Define parameters for selected history.*

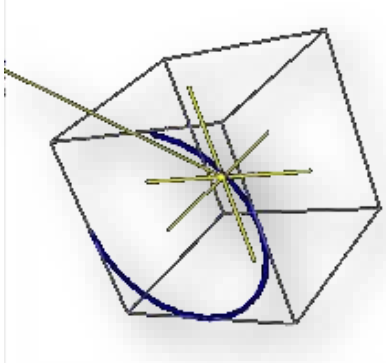
5. *Define time interval for selected history*



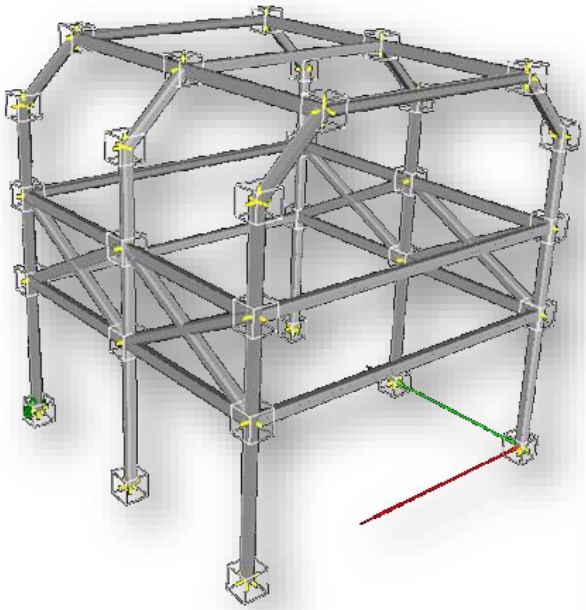
On the Solution 3D view the defined Force will be illustrated by a blue arrow.

Moments assigned to a Node will be illustrated by blue round line (one for each X, Y and Z d.o.f).



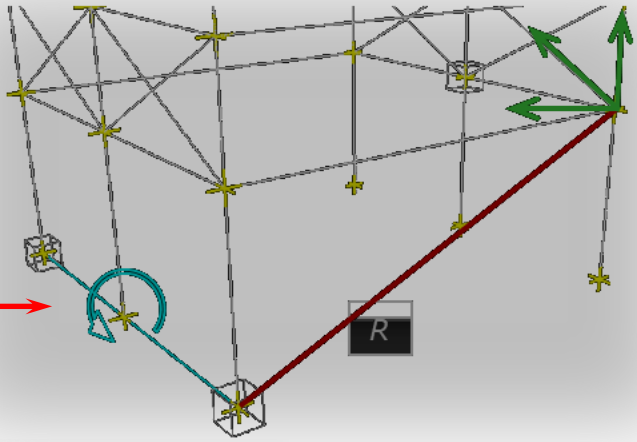


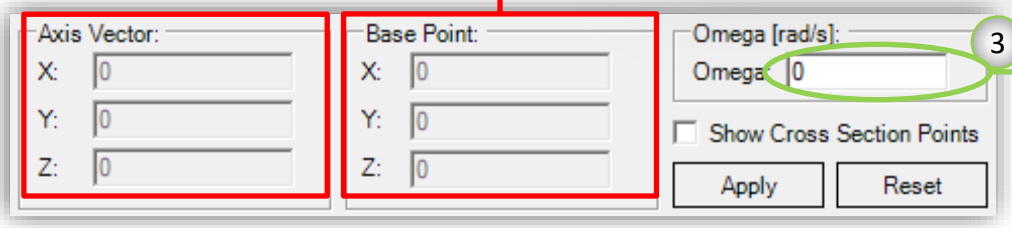
"Assign Initial Velocity" tool



1) *After choosing the **"Assign Initial Velocity"** tool from the main toolbar select all nodes to which you wish to assign initial velocity.*

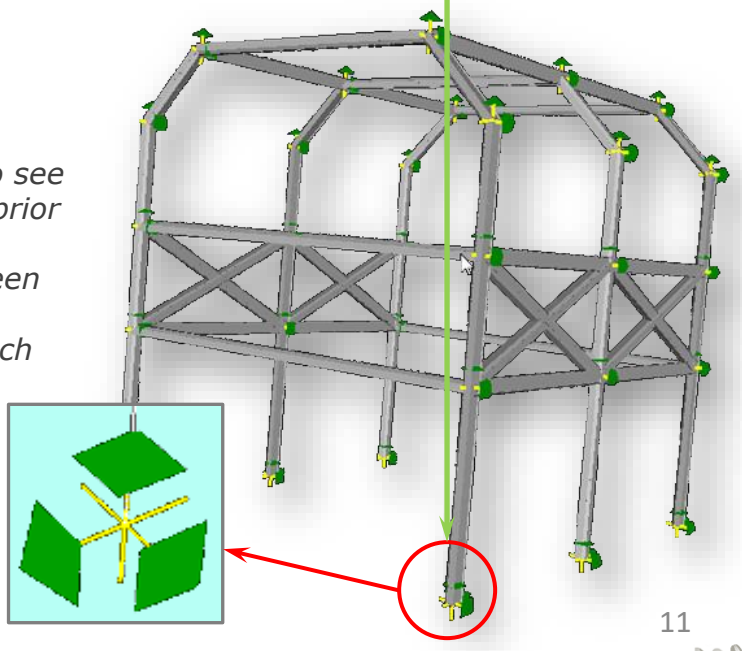
2) *Select two Nodes in the 3D view in order to define the rotation axis. You can also enter the coordinates manually*



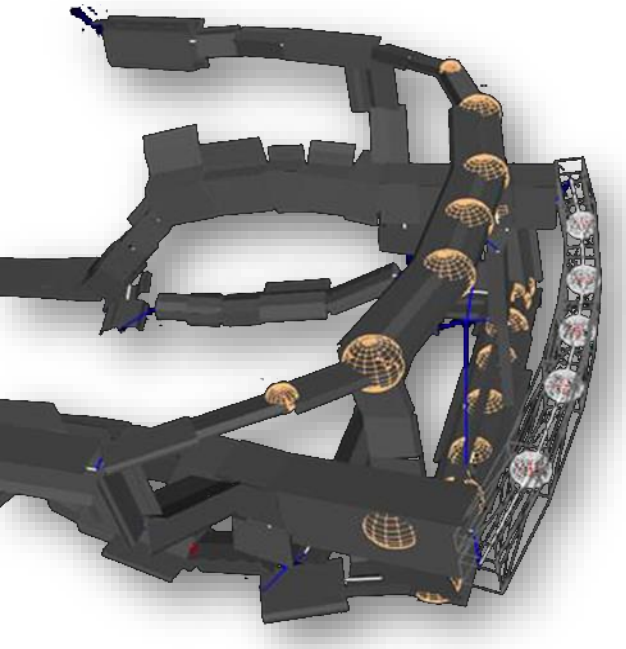


3) *Define the angular velocity*

In the VCS 3D view you will be able to see set of linear velocities assigned to all prior selected Nodes.
Linear velocity is represented by 3 green arrows.
You can view assigned velocities of each Node in its Properties window.



CONTACT SETTINGS – BASIC INFORMATION



Definition of contact settings starts when the whole model structure is prepared.

The contact/impact can be simulated for all structural VCS elements and all their combinations. The only exception is the contact between deformable blocks of moving and stationary barriers.

The VCS implements specialized contact detection routines and wholly physical contact mechanics. The general contact detection routines, used e.g. in FE programs, are a CPU demanding processes that would take more CPU time then calculation of mechanical response of Super Elements model.

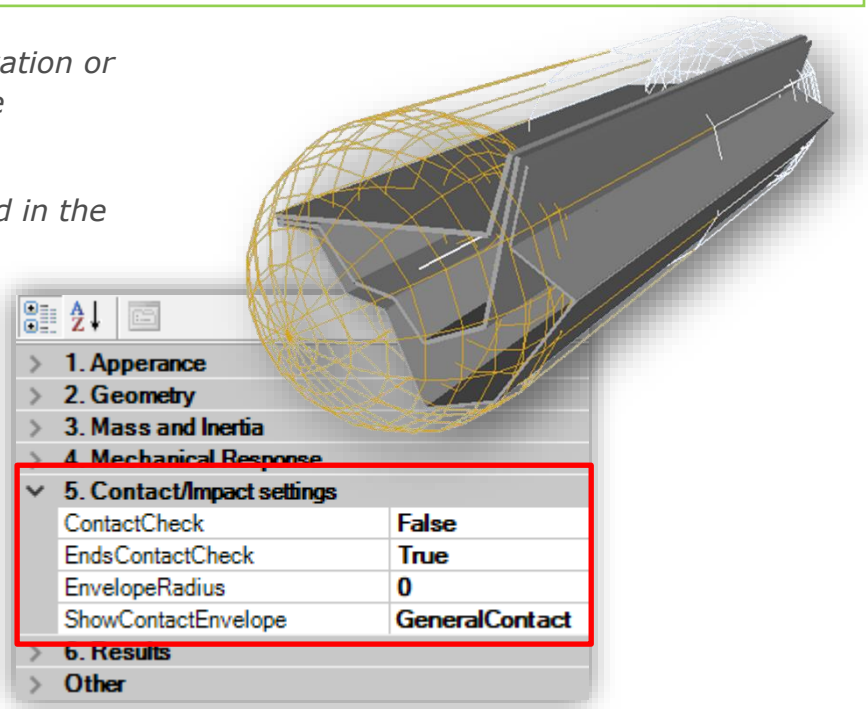
*In order to keep the CPU time efficiency, the VCS contact/impact algorithms use a multi-level contact detection procedures based on the **Contact Envelope concept**. Most of the CPU time can be saved by sparse specification of possible **Contact Pair.***

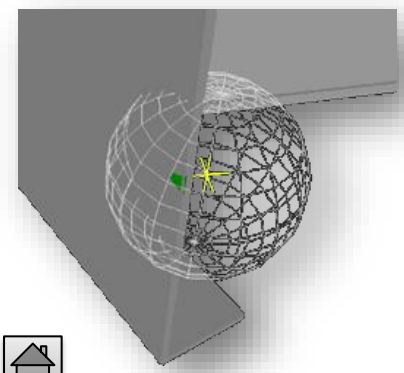
*In VCS the **Contact Settings** are defined in two steps described below:*

Step 1 – create contact bodies

*The first step includes the activation or creation of contact bodies – the **CONTACT ENVELOPES.***

Contact Envelopes are activated in the object's Properties window.





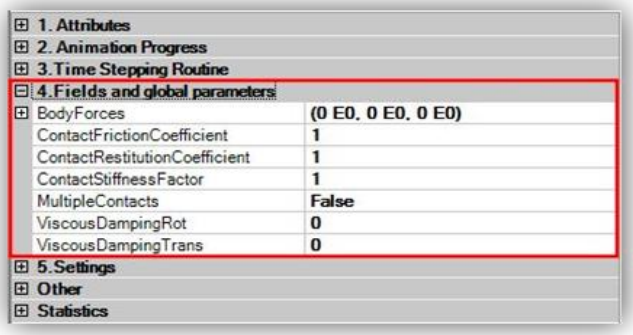
*Contact Envelope can be also created as a **mass-less Rigid Body**.*

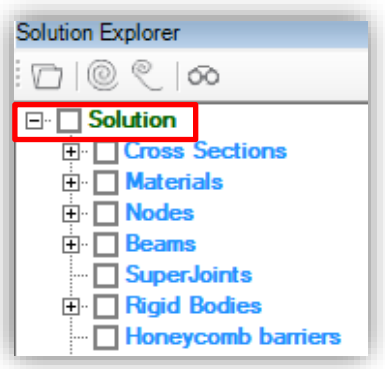
CONTACT SETTINGS – BASIC INFORMATION

Step 2 – contact definition

At this point the User needs to choose between two methods of ***CONTACT DEFINITION.***

- ***GENERAL CONTACT*** *- allows detection of global contact between all the elements with enabled 'Contact Check' property (switch position has to be set to 'True').*
  Note that in case of General Contact the definition of objects contact response is made in the 'Solution Settings' properties window





- ***CONTACT PAIR*** *- Contact Pair is defined between two selected elements. This method gives wider range of settings. Apart from the basic parameters (contact friction coefficient, contact restitution coefficient and contact stiffness factor) the Contact Law can be defined.*

IMPORTANT PRACTICAL ADVICES FOR CONTACT DEFINITION

- *Use contact spheres rather than contact envelopes whenever possible*
- *Avoid using box-to-box contact pairs (detection of edge-to-edge contact is sometimes ambiguous)*
- *Define as little contact pairs as possible.*
- *The contact between box and cylinder is detected only at the edges of the box.*
- *By default, collision of Rigid Bodies in VCS is assumed to be PERFECTLY ELASTIC (restitution coefficient equals 1)*

CONTACT ENVELOPES

VCS contact/impact algorithms use a multi-level contact detection procedures based on the Contact Envelope concept.

The contact detection routine is multi-level. When the Contact Check switch is on the object is embedded in a spherical envelope. The contact is then checked for the spherical envelopes of all elements. Once this first-level contact is detected the second-level contact routine checks the contact with more refined contact envelopes. The shapes of these envelopes depend on the geometry of a corresponding object.

- *The simplest **Contact Envelopes** are defined for **Rigid Bodies** – they simply coincide with the shape of a rigid body.*

- *The **Contact Envelope of a Beam** element consists of a cylindrical surface, which is used for detection and handling of an impact against side faces of the beam. In addition two spherical envelopes can be added at both ends of the cylindrical envelope in order to control end impacts of the beam.*

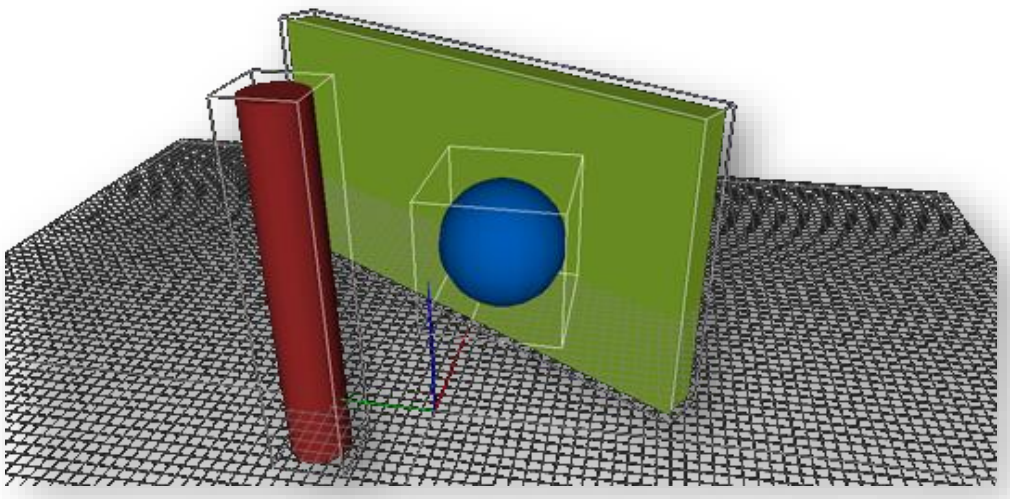
- *Note that separate Contact Envelope is defined for **Deformable Barrier** contact. This envelope is generated automatically when contact with barrier deformable block is detected.*

RIGID BODIES AS CONTACT ELEMENTS

Contact Envelopes can be defined for Rigid Bodies – they simply coincide with the shape of a Rigid Body. In VCS contact is detected between three basic 3D primitives: spheres, cylinders, boxes (parallelepipeds) and all possible combinations of the above.

The usage of mass-less Rigid Bodies as contact elements is the most effective means of contact modeling.
Among all the rigid types implemented in VCS the spherical body is most CPU effective in contact modeling. Second in line is the contact between cylinders and finally the most CPU demanding is the contact algorithm for boxes.

IMPORTANT
Note that by default collision of Rigid Bodies in VCS is assumed to be PERFECTLY ELASTIC (restitution coefficient equals 1)



CONTACT ENVELOPES

VCS contact/impact algorithms use a multi-level contact detection procedures based on the Contact Envelope concept.

The contact detection routine is multi-level. When the Contact Check switch is on the object is embedded in a spherical envelope. The contact is then checked for the spherical envelopes of all elements. Once this first-level contact is detected the second-level contact routine checks the contact with more refined contact envelopes. The shapes of these envelopes depend on the geometry of a corresponding object.

- *The simplest **Contact Envelopes** are defined for **Rigid Bodies** – they simply coincide with the shape of a rigid body.*

- *The **Contact Envelope of a Beam** element consists of a cylindrical surface, which is used for detection and handling of an impact against side faces of the beam. In addition two spherical envelopes can be added at both ends of the cylindrical envelope in order to control end impacts of the beam.*

- *Note that separate Contact Envelope is defined for **Deformable Barrier** contact. This envelope is generated automatically when contact with barrier deformable block is detected.*

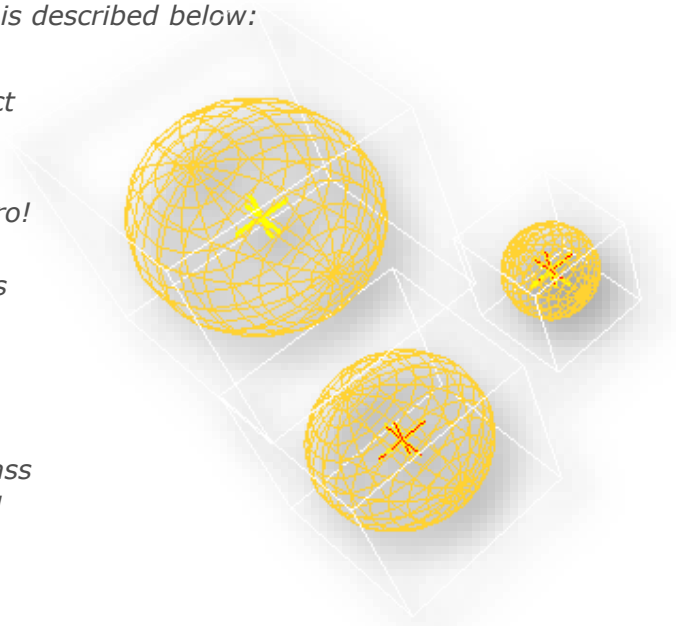
SPHERICAL CONTACT INTERFACE

The contact interface can be defined by a mass-less sphere. . Among all the rigid types implemented in VCS the spherical body is most CPU effective in contact modeling.

The procedure of defining such contact interface is described below:

1. *A sphere (Rigid body) is defined on the impact Node*

2. *The Mass of the sphere needs to be set to zero! (massless contact interface).*
   *Note that the **'Lock Density'** flag in the Mass and Inertia section of the Properties window needs to be set to **'False'.***



Otherwise, VCS will automatically keep hold of density during resize operations and will treat Mass as a dependent variable. In consequence manual changes of the mass property will be impossible.

3. *In order to run the VCS contact / impact processor on any object of the model the Contact Check flag must be set to 'TRUE' in an appropriate object Properties window.*
   In order to view the spherical envelope, the Show Contact Envelope flag must be set to 'True'.

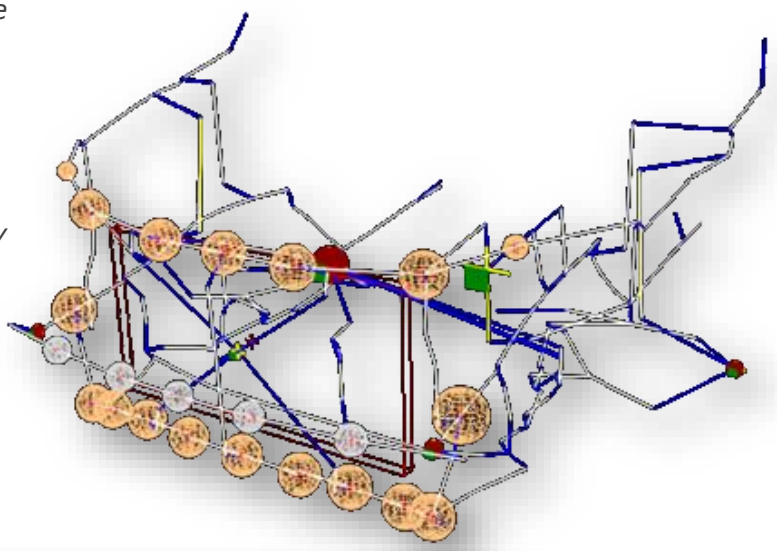
CONTACT SETTINGS – CREATE CONTACT BODIES

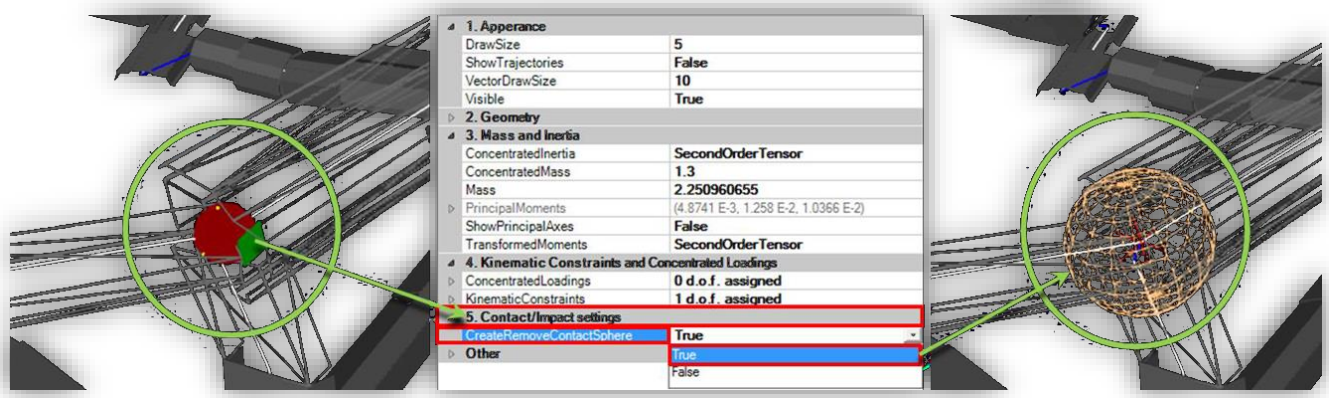
The contact interface can be defined by a mass-less Rigid Body.
Among all the rigid types implemented in VCS the spherical body is most CPU effective in contact modeling.
A sphere (Rigid Body) is defined on a selected impact Node.

After selecting an appropriate Node go to the Contact / Impact Settings section of the Properties window (as illustrated in the picture below).

Activation of contact properties for Node is available by switching ***Create-Remove Contact Sphere*** *flag to* ***'True'.*** *The property change results in creation of Rigid Body (Sphere). The sphere is massless Rigid Body with default radius value.*

The radius value can be adjusted when a Rigid Body is selected on 3D or Solution Explorer. It is important to define the radius to assure it refers to the dimensions of the super beam.





CONTACT BOX & CONTACT CYLINDER

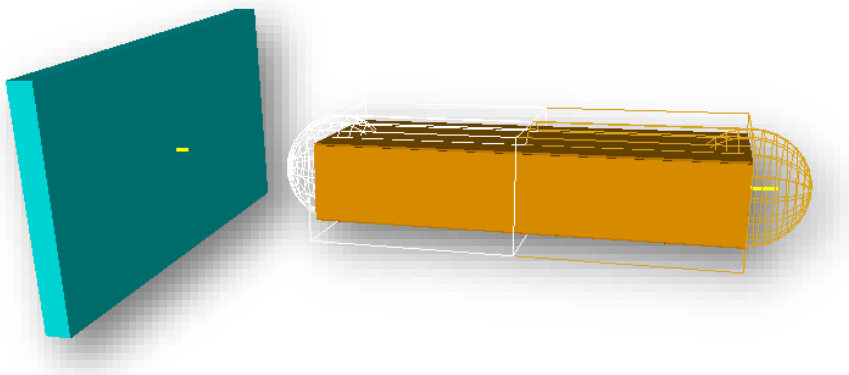
Similarly to the spherical contact, a contact envelope can be created for boxes and cylindrical Rigid Bodies.
The procedure of creating a contact envelope for boxes and cylinders is nearly identical as the one described for cylindrical contact
The mass of the Rigid body needs to be set to zero.
Note that the 'Lock Density' flag in the Mass and Inertia section of the Properties window needs to be set to 'False'

In order to run the VCS contact / impact processor on any object of the model the Contact Check flag must be set to 'TRUE' in an appropriate object Properties window.
In order to view the spherical envelope the Show Contact Envelope flag must be set to 'True'.
In case of cylindrical Contact Envelops the Ends Contact Check flag needs to be set to 'True' in order to properly detect the contact during simulation.

CONTACT ENVELOPES FOR BEAM ELEMENTS

In VCS the default Beam Contact algorithm uses the concept of Contact Envelopes. The envelope consists of cylinder and sphere. Both surround one half of the beam element together with corresponding node.

The default radius of the envelope is defined as an average width of cross section and can be modified by the user for specific contact models.

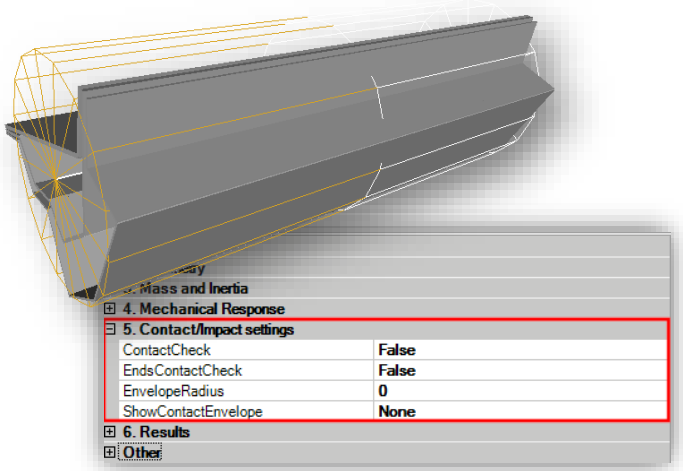


The Contact Check switch and controls of the envelope type are collected in the Contact/Impact setting section of the Beam's Properties Window.

The cylindrical surface of the Contact Envelope is used for detection and handling of an impact against side faces of the Beam.

In order to create a cylindrical Envelope, you need to set the "Contact Check" flag to 'True'

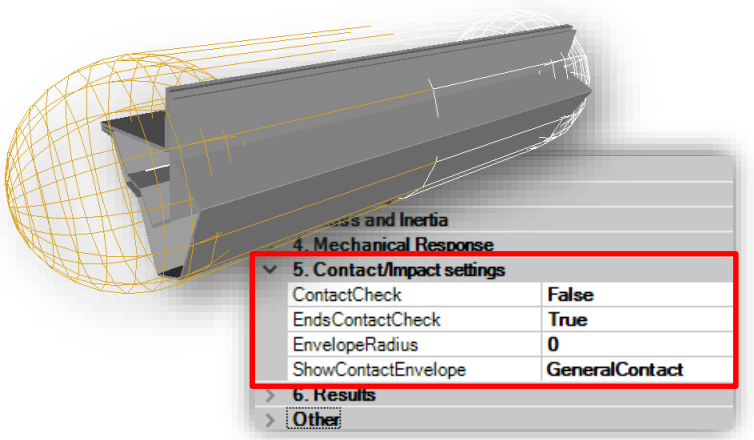
When the Contact Check is set to True automatically Show Contact Envelope will be switched to General Contact.



In addition, two spherical envelopes can be added at both ends of the cylindrical envelope in order to control end impacts of the beam.

In order to create the spherical Envelopes, you need to set the "Ends Contact Check" flag to 'True'

The Envelope radius value is automatically set to 0 [mm], which corresponds to Default Radius Envelope.

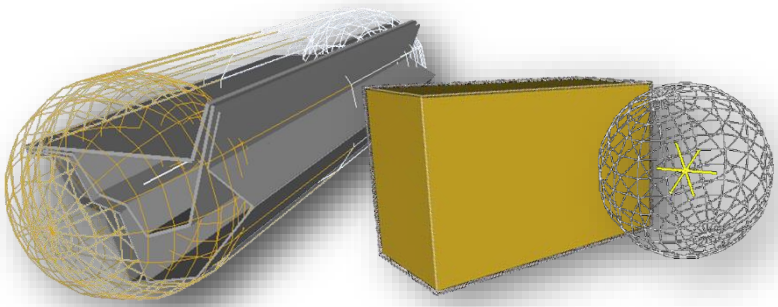


Contact Pairs Definition – Step By Step

Contact Pair *is a user-predefined set of two elements that may be engaged in contact during the simulation process. The contact is* ***exclusive*** *– it is tested only between objects bind into a Contact Pair.*
Definition of contact pairs significantly reduces CPU time in the case of large number of contact elements. In addition contact response of paired elements can be defined by the user for each pair individually.

Contact Pairs may include:

- *Rigid Bodies and/or*
- *Contact Envelopes.*



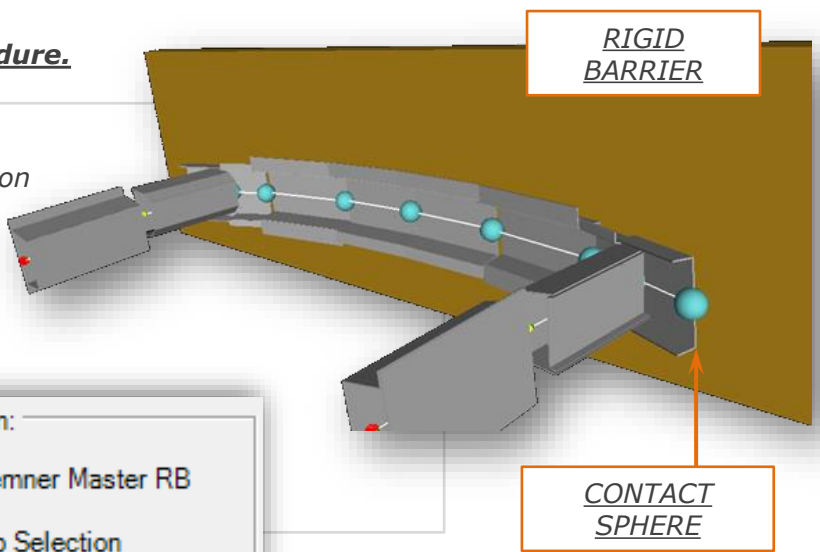
Contact Pair

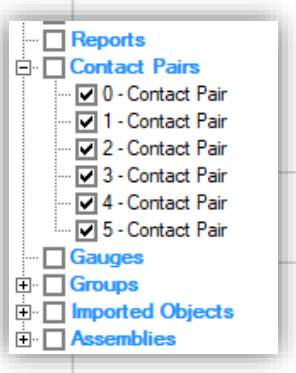
"Create Contact Pair" tool

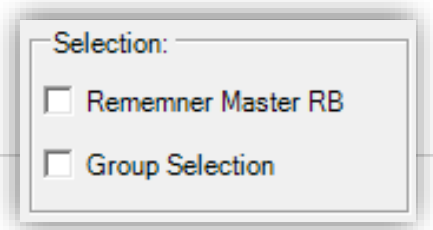
The Contact Pair method of contact definition enables the user to define specific contact laws between two selected objects.

Create Contact Pair – basic procedure.

Contact Pair is defined on 3D Solution views by clicking in turn on two elements with active Create Contact Pair tool
Contact Pairs may include:
- *Rigid Bodies and/or*
- *Contact Envelopes.*



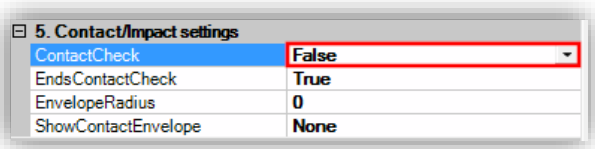




The manually defined Contact Pairs are catalogued under the Contact Pairs branch of the Solution Explorer tree.

Important Notice

The ***Contact Check*** *switch should be set to* ***'False'*** *for each object when contact pair is defined.*



"Create Contact Pair" tool

Contact Pair – Properties.

1. Contact Elements	
FirstObject	Contact sphere for 1- Node
SecondObject	Contact sphere for 2- Node
ShowContacEnvelopes	False
2. Contact Law	
SelectedContactLaw	Standard
2a. Standard Contact	
FrictionCoefficient	0.1
RestitutionCoefficient	1
StiffnessFactor	1
2b. User Defiend Contact	
ContactLaw	
3. Run Time Contact Data	
ContactStiffness	-1
ContactTime	-1
Other	
Layer	Default
Name	1 - Contact Pair

*Constitutive law for contact/penetration is controlled by the **Selected Contact Law** switch of the **Contact Pair's Properties window**.*

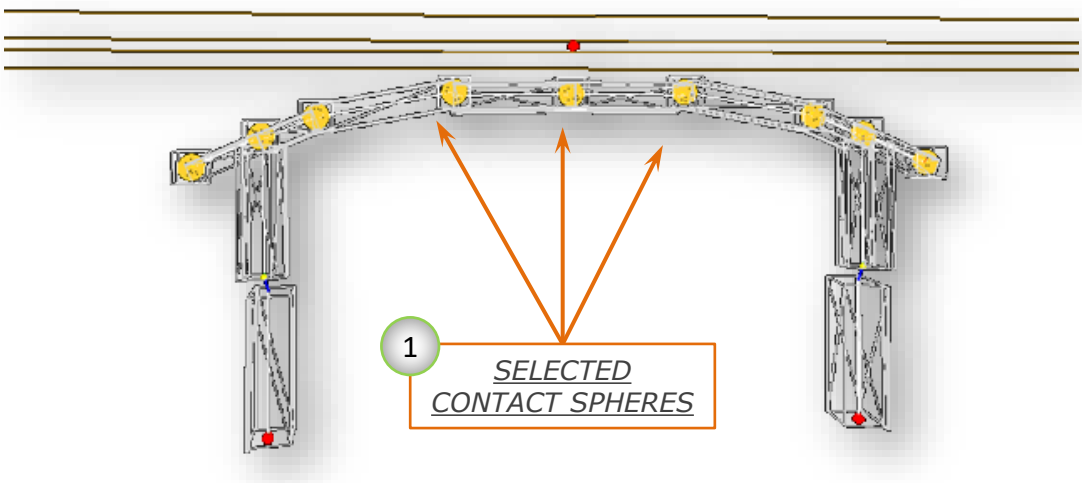
Standard law uses suitably modified parameters of general rigid-body like impact mechanics (restitution coefficient, elastic stiffness etc.).

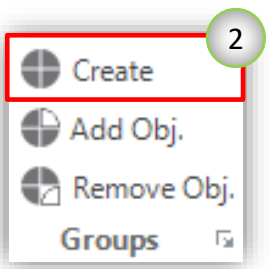
The User Defined option allows for definition of the contact law by means of a 2D characteristic. The characteristic must define positive penetration resistance as a function of positive penetration.

*The method of contact checking depends also on the global settings at the level of Solution. In the **Multi Contact** field of the **Solution's Properties window** the flag needs to be set to **'False'.** The contact will then be tested by pairs of objects that get firstly into contact during the simulation*

Create Contact Pair – Group Selection option.

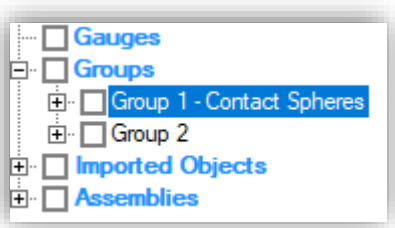
With the usage of the "Group Selection" option available in the tool Settings window you can create number of contact pairs between a contact objects and number of contact objects included previously in a Group.



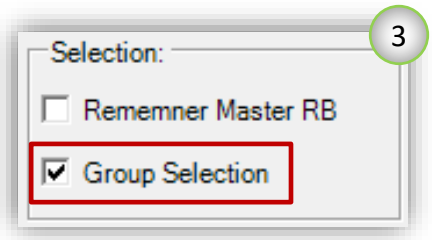


In the Solution 3D view select all required contact bodies. Afterwards click on the "Create Group" tool available in VCS main toolbar.

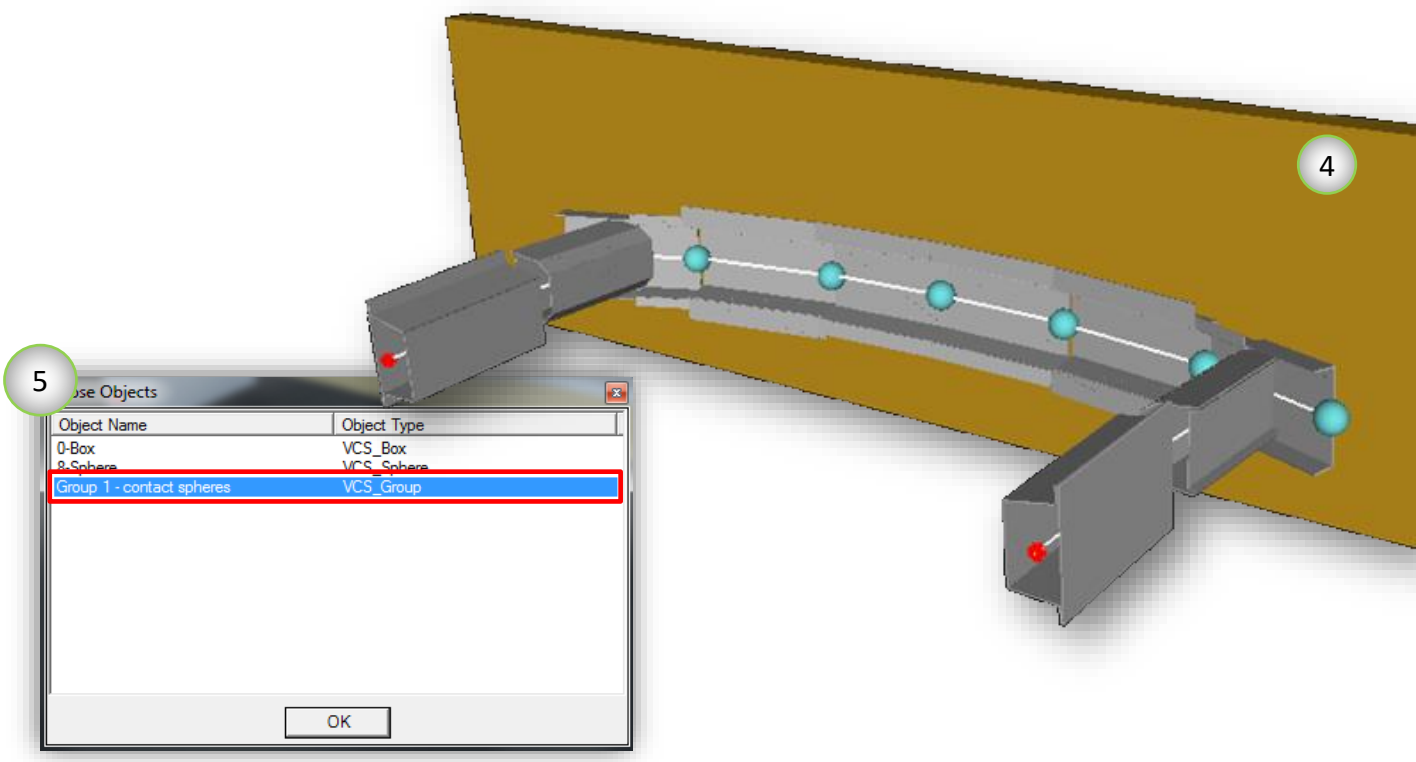
All selected objects will be added to a newly created Group. The Group can be found in the Solution Explorer tree.



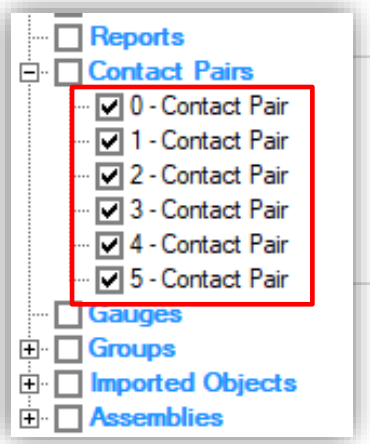
"Create Contact Pair" tool



Select the "Create Contact Pair" tool from the VCS main toolbar.
In the Tool Settings window click on the "Group Selection" option.



In the Solution 3D view select the Master contact object (box rigid barrier in case of the presented example).
Afterwards select the previously defined Group of contact objects.



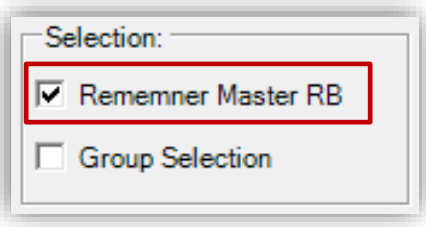
Number of Contact Pairs will be automatically created and added to the Solution Explorer tree.

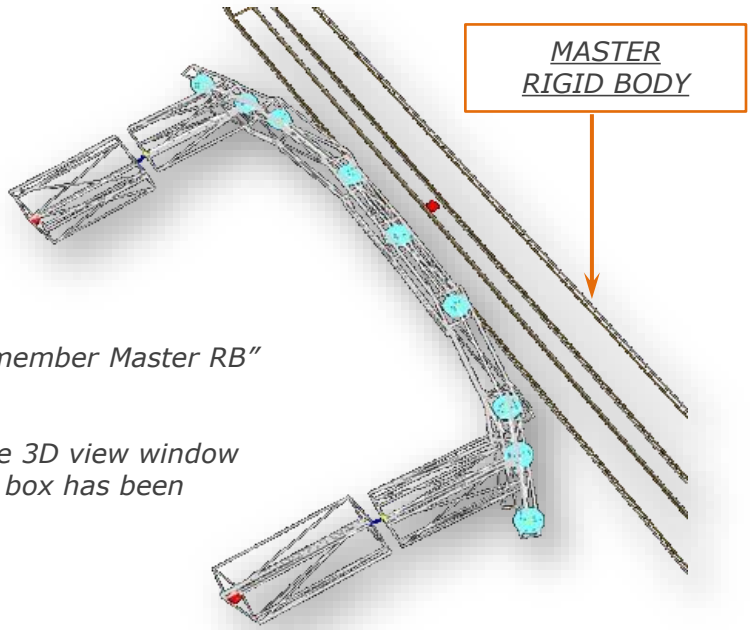
The newly created contact pairs will include all object present in the previously defined Group.

"Create Contact Pair" tool

Create Contact Pair – Remember Master Rigid Body option.

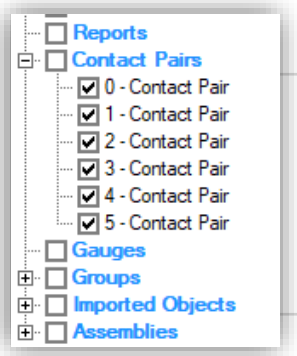
With the usage of the "Remember Master Rigid Bode" option available in the tool Settings window you can easily create number of Contact Pairs which include the same master Rigid Body. In the presented example the box barrier is defined as the master rigid.





In the Tool Settings window select the "Remember Master RB" option.

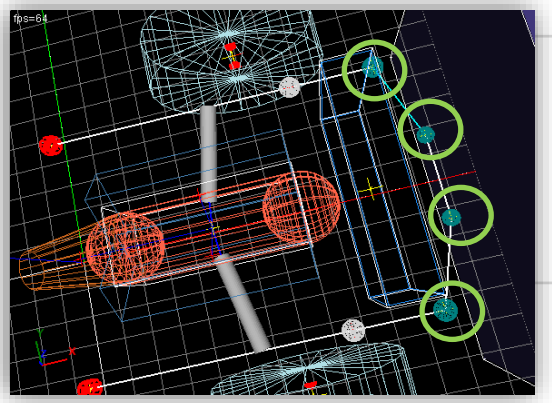
Afterwards click on chosen Rigid Body in the 3D view window (in case of the presented example the rigid box has been selected)



Now click in turn on selected contact objects in order to create number of Contact Pairs (in case of the presented example, contact spheres were selected.

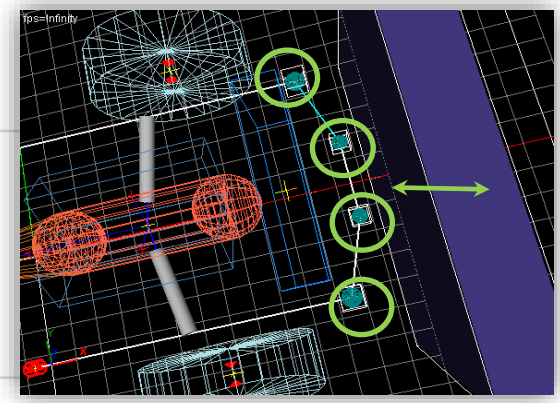
Newly created contact pars can be found in the Solution Explorer window.

CONTACT SETTING IN ELEMENTARY MODEL - EXAMPLE



STEP 1:

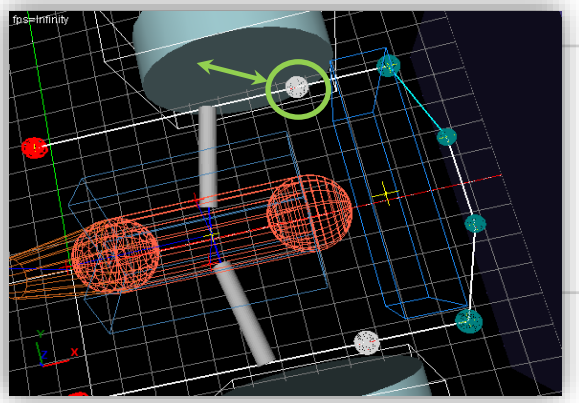
Contact spheres are created on bumper beams' nodes (Node's properties window).
Additional contact sphere is defined for the longitudinal beams



STEP 2:

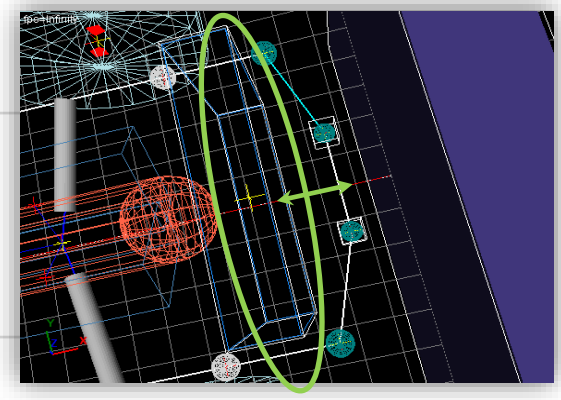
Bumper – barrier contact pairs are defined.

Contact pair is created between each contact sphere of the bumper assembly and the barrier rigid box.



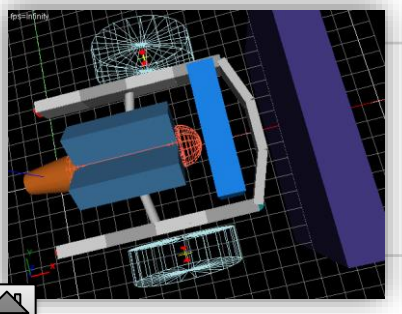
STEP 3:

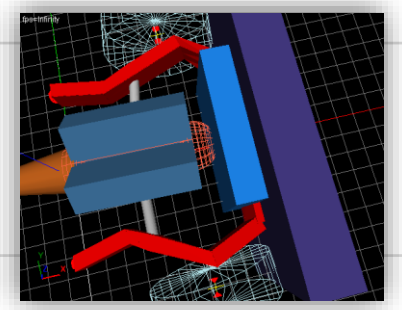
Contact pairs are defined for contact spheres of the longitudinal beams and wheels (rigid cones) – it is assumed that bending longitudinal elements will intrude wheels during the simulation



STEP 4:

Bumper – radiator contact pair is created. It is assumed that the radiator rigid will be in direct contact with the rigid barrier.



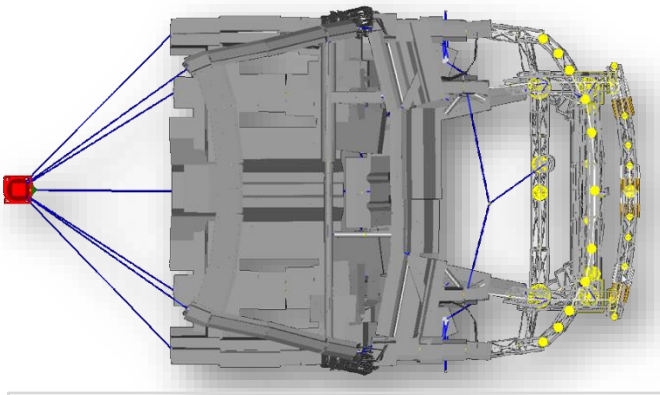


Complete model with fully defined contact

and

Results of the simulation

CONTACT ZONES

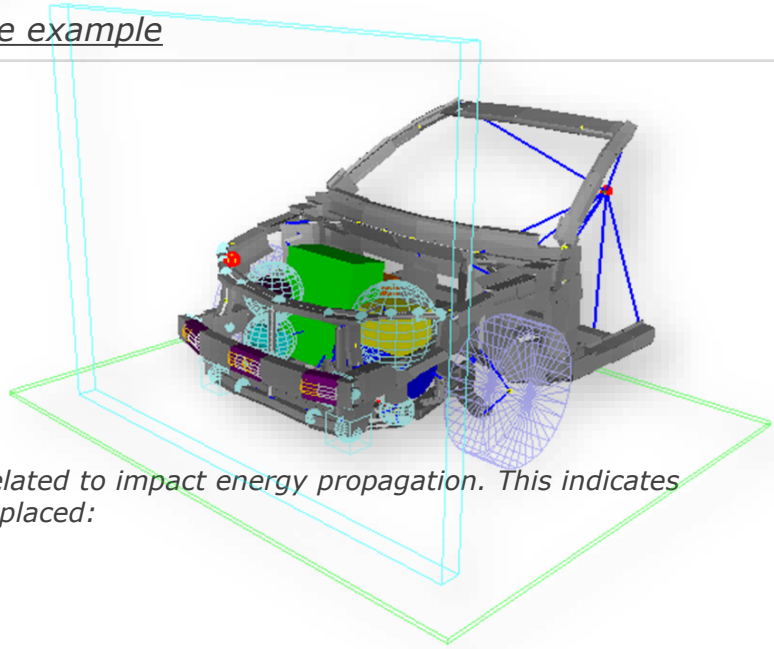


Without definition of rigid bodies and contact pairs elements of the structure WILL NOT DETECT each other.

In order to reflect the contact between further elements of the structure, for instance engine and S-Frame, additional rigid bodies and contact pairs must be created.

Contact Zones – front impact case example

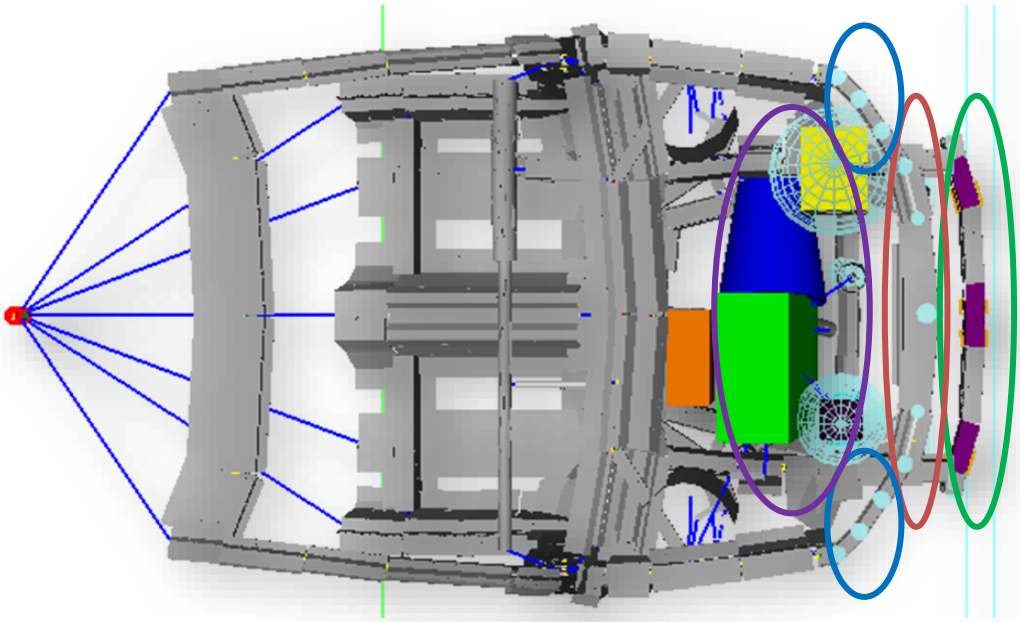
1. *Vehicle model*
2. *Rigid wall*
3. *Impact velocity: 56 km/h*
4. *Simulation time: 90 ms*



The model can be divided into zones related to impact energy propagation. This indicates where the contact elements should be placed:

1. *Bumper zone*
2. *Radiator zone*
3. *Upper Rail zone*
4. *Engine and suspension frame zone*

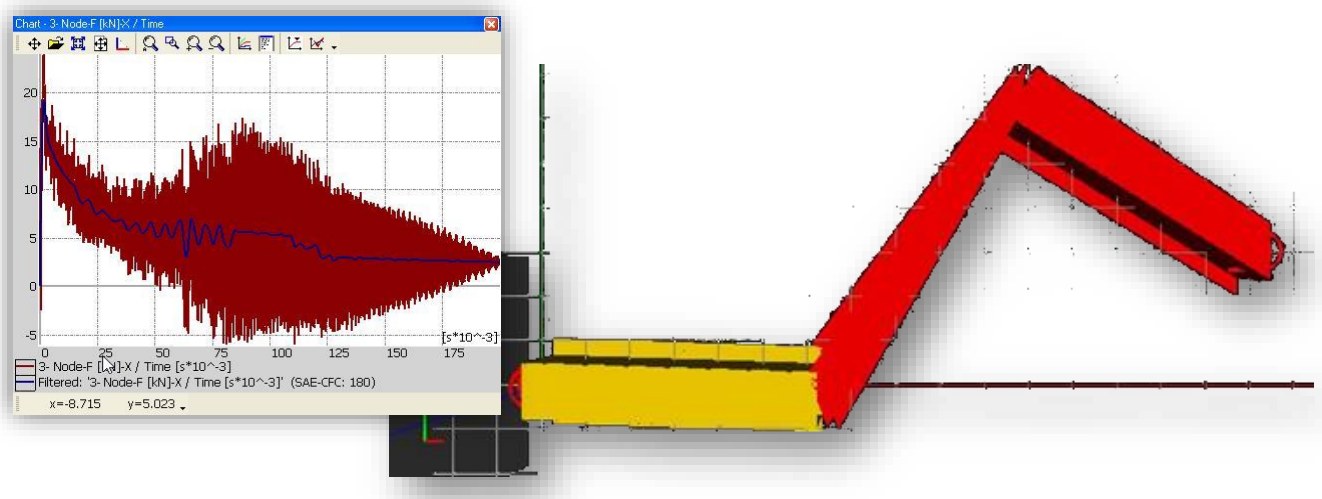
When creating contacts, it is recommended to set the solution settings at the beginning and check the animation and deformation state from time to time to find out where the contacts are needed. Contact pairs should be placed where the interpenetration of objects occurs. Observe if energy has been dissipated and if further contact zones are needed.



CONTACT PAIRS SETTINGS - EXAMPLES

S – FRAME SIMULATION – Rigid Bodies contact

The description of contact definition between Rigid Bodies is based on S – Frame simulation.



The reference model of S – Frame consists of three Super Beam Elements with 500 kg mass attached at the far end traveling with constant velocity of 1 [m/s].

In the VCS reference model the impacting end of the frame is fixed which corresponds to raw modeling of contact without sliding named as "sticky contact". The reference time history for this model is the x component of axial crushing force measured at far end of the frame

(*) STICKY CONTACT

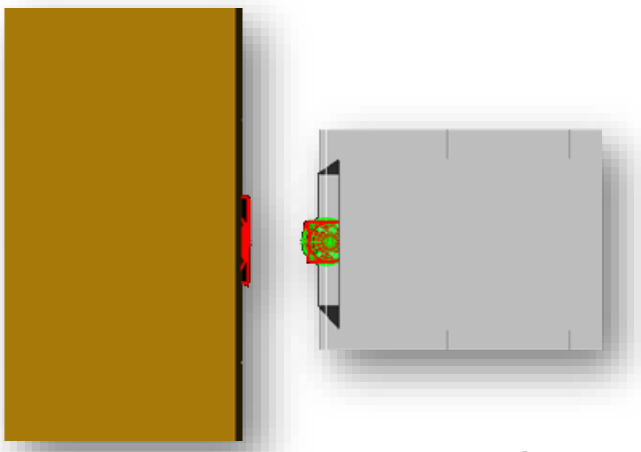
The sticky contact is understood as a contact in which the contacting pair does not slide on each other as long as the normal contact force exists. In other words, the contact node can only penetrate the barrier in normal direction.

The sticky contact with the use of rigid bodies contact interface is modeled by imposing fixed conditions on all rotations and two transitions of the impacting End Node (for example y,z)

S – FRAME - Spherical contact - example

Example below is considered as first followed by more realistic modeling of frame-barrier contact event with box-like interface application and standard contact envelopes of beam elements.

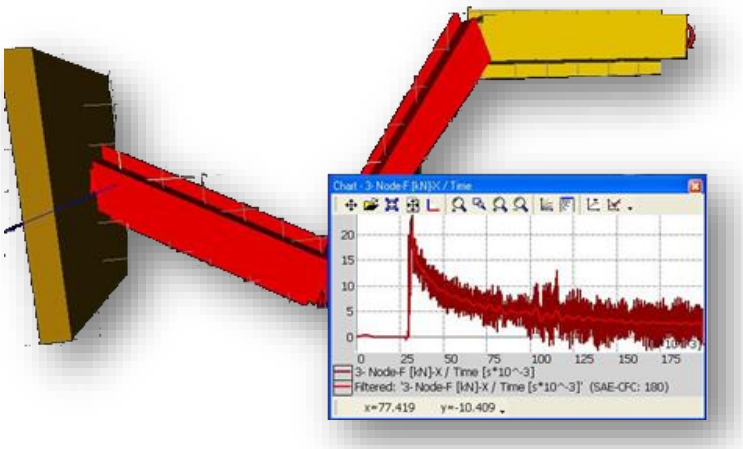
In the first example a Sphere is defined as a contact interface at the impacting Node of S Frame. The mass of the sphere is set to zero. The sticky contact condition is modeled by imposing fixed boundary conditions on all rotations and translations in Y and Z direction of the impacting node. Initial boundary conditions are defined at the node with concentrated mass.



CONTACT PAIRS SETTINGS - EXAMPLES

In order to take into account contact impact behavior in VCS calculations the Contact Check flag must be with respect to General Contact or Contact Pairs Procedure.

A result of S frame crushing equipped with spherical contact interface is illustrated on the right.
The chart of force in time function shows that the construction hits the barrier in about 30 ms. Crushing characteristic is very much similar to the reference calculations however small differences are caused by different deformation mode.

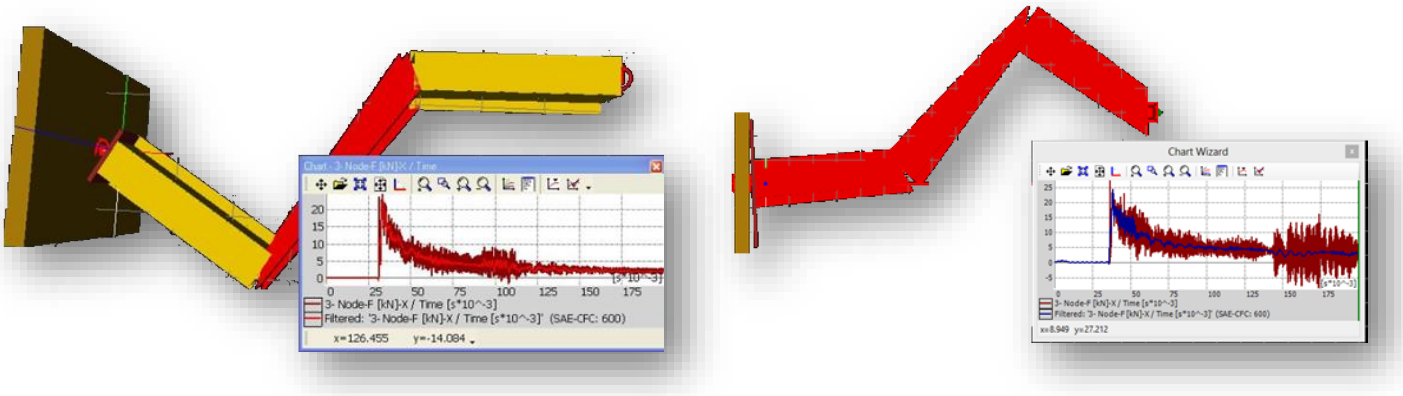


Different deformation mode results from proper handling of elastic wave traveling from the impacting end of the frame. In the reference model elastic loading wave travels from the far end of the frame where constant velocity is applied.

S – FRAME - Box-type contact interface

In this example the rotation constraints at the impacting end can be removed since the interaction of box-like padding (Rigid Box) of the impacting end controls bending response of the beam. The translation constraints in y and z directions are still retained in order to model the sticky contact type.

Note that in this case the bending response of the impacting end of the frame depends on the size of front pad.



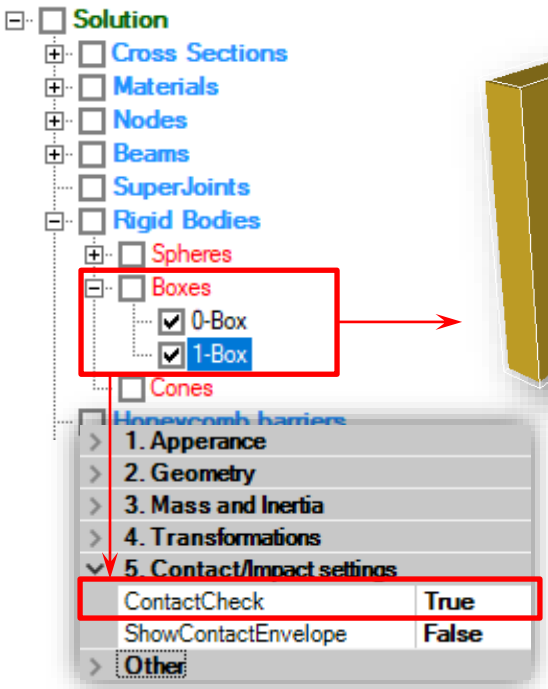
Results of impact simulation for different sizes of end padding

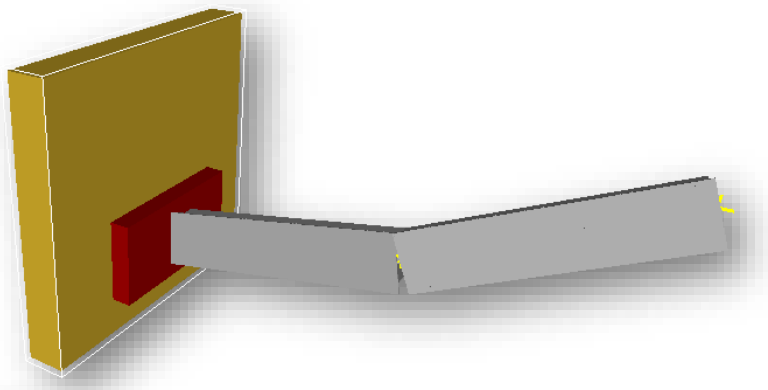
Important Note:

The user can define combinations of contact between Spheres, Boxes and Cylinders. The recommendation regarding contact definition between cylinders and boxes is to switch the "End Contact" to 'True' position.

GENERAL CONTACT

GENERAL CONTACT *- allows detection of global contact between all the elements with enabled* ***'Contact Check'*** *property (switch position has to be set to* ***'True'****).*

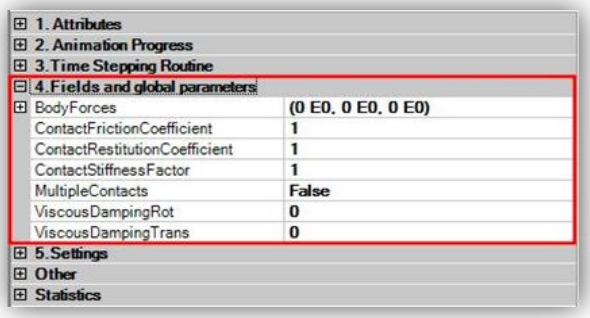




When the General Contact method is selected all contacts (objects with the 'Contact Check' flag set to 'True') within the Solution are governed by the same contact law defined in the Property window of the Solution.

Additional settings related with General Contact definition are located in the Solution's Properties window (top most Object of the Solution Explorer tree).

In the 'Fields and Global Parameters' section of the Solution's Properties window you can find settings for:



- ***Contact Friction Coefficient*** *– coefficient of friction in collision calculations.*
- ***Contact Restitution Coefficient*** *– defines characteristics of collision process between rigid bodies in the model. Value 0 gives perfectly plastic response, Value 1 purely elastic response.*
- ***Contact Stiffness Factor*** *– Reduction factor of default local elastic stiffness in contact problems. The default value is the highest stiffness that guaranties stable result.*
- ***Multiple Contact*** *- switches on all possible contact envelopes and checks the contact between each element in 3D structure*

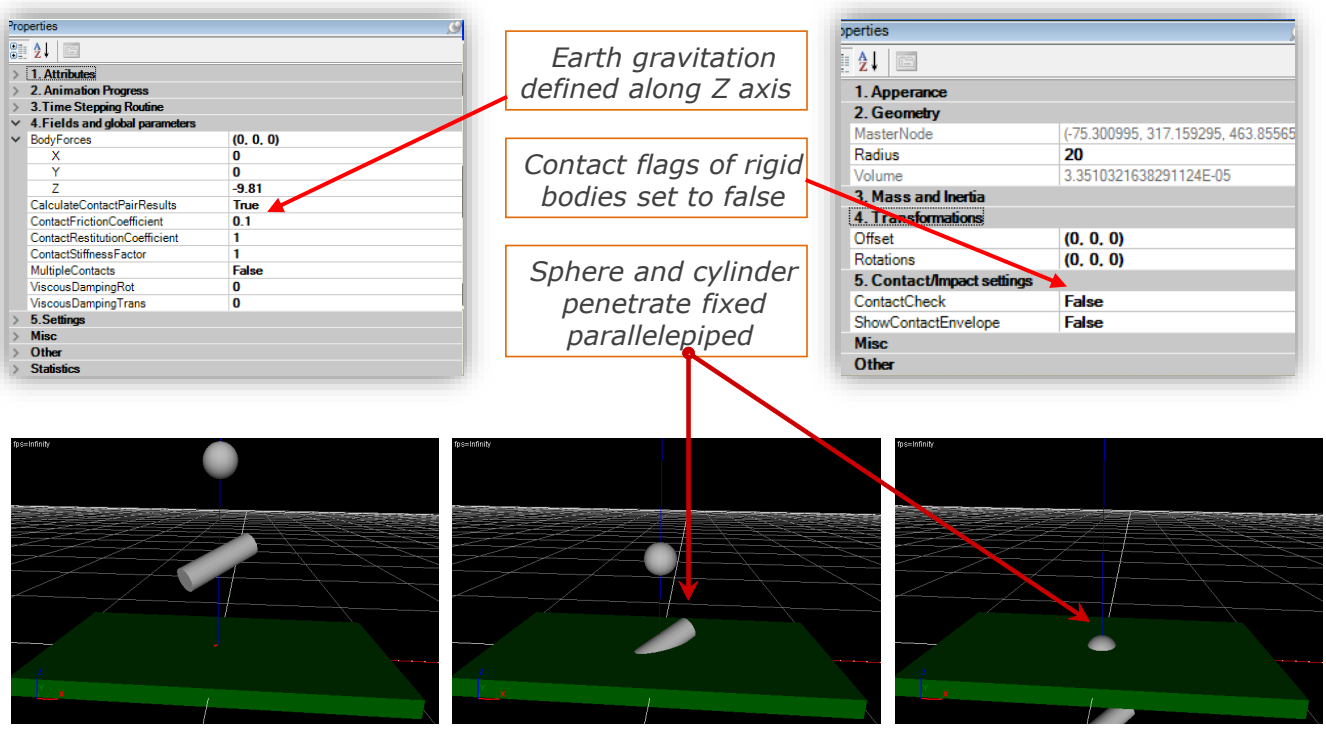
In case of General Contact method the Multiple Contact flag needs to be set to 'True'. This assures that the contact will be checked between all the collision candidates regardless of the contact sequence (inclusive contact).

Note that the Multiple Contact stings are the most CPU time demanding. In addition General Contact Does not allow for definition of specific contact response between selected objects. For this reasons General Contact option of contact settings is rarely used in automotive applications.

GENERAL CONTACT – SETTINGS EXAMPLE

When the general contact method is selected all contacts within the model are governed by the same contact law defined in the property windows of the Solution, (left property window in the picture below).

In the elementary example, two rigid primitives sphere and cylinder fall down onto the box-shaped (parallelepiped) ground. With default VCS settings, when there are no contact conditions defined both primitives penetrate stationary ground and continue free fall. The property windows in explain VCS settings corresponding to this case.

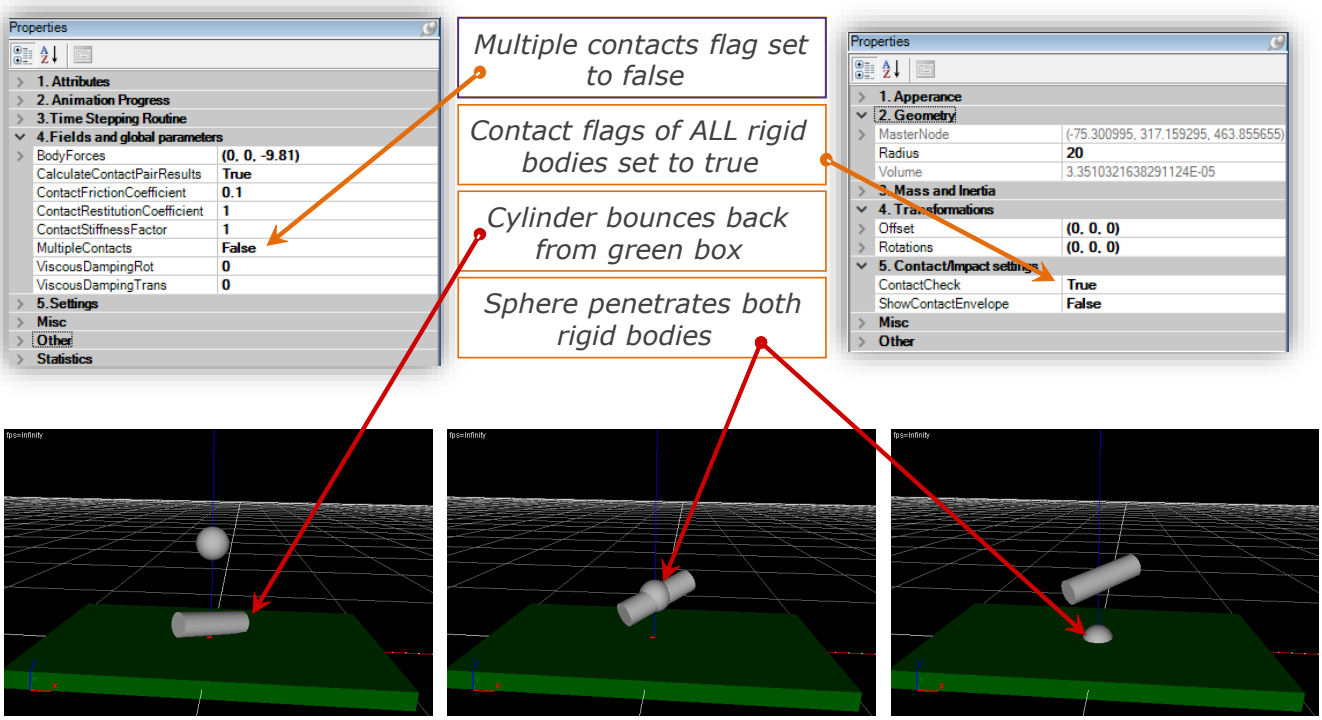


The contact is checked between Rigid Bodies (and derived objects) when the local Contact Check flag is set to 'True' in all objects for which contact is to be tested during simulation. The method of contact checking depends also on the global settings at the level of Solution object. If the Multiple Contacts flag is set to 'False' the contact is tested by pairs of objects that get firstly into contact during the simulation.

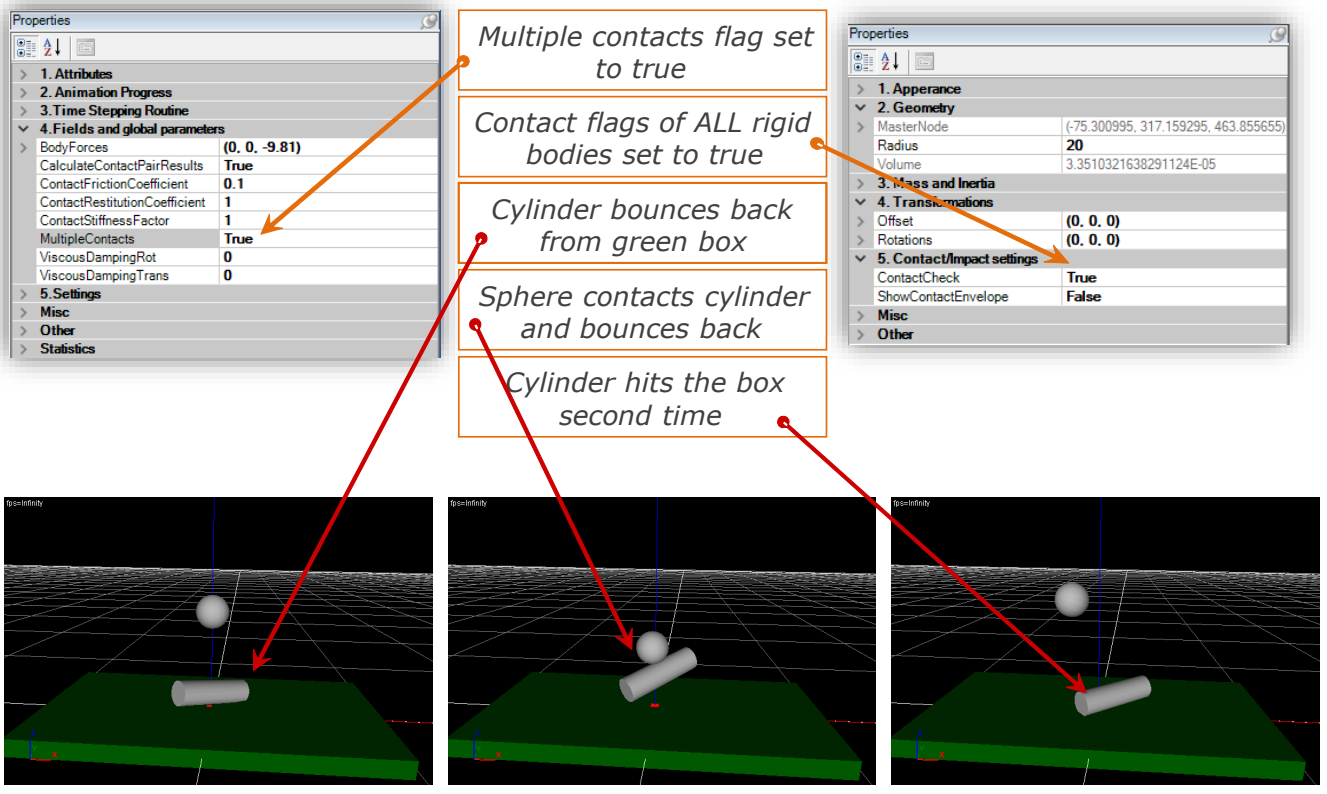
In other words when two objects are bind into contact pair the contact between these bodies and remaining potential contact candidates is not tested (exclusive contact condition). These settings are illustrated in the picture below for the free fall example discussed earlier.

When cylinder hits the ground the box and cylinder are bind into an exclusive Contact Pair. The cylinder bounces back but contact is no longer tested between ground/box pair and other collision candidates (in this case sphere). Consequently, during the rebound motion cylinder penetrates the sphere. Later sphere penetrates the ground.

GENERAL CONTACT – SETTINGS EXAMPLE



When the Multiple Contacts flag is set to 'True' the contact is checked between all the collision candidates regardless of the contact sequence (inclusive contact)



With these settings the discussed case of free fall follows our physical intuition: first cylinder bounces back from the ground hits the sphere, both objects change trajectory etc. This scenario of a Global Contact setting is illustrated in the picture above. It is obvious that the Multiple Contact settings are the most CPU time demanding. In addition, General Contact does not allow for definition of specific contact response between selected objects. For these reasons, the general contact option is rarely used in automotive applications.

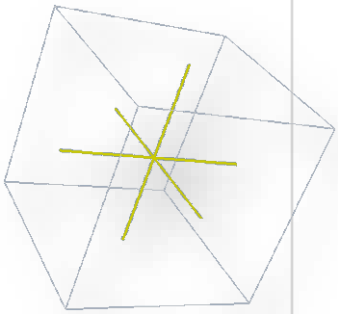
"Create Deformable Barrier" tool

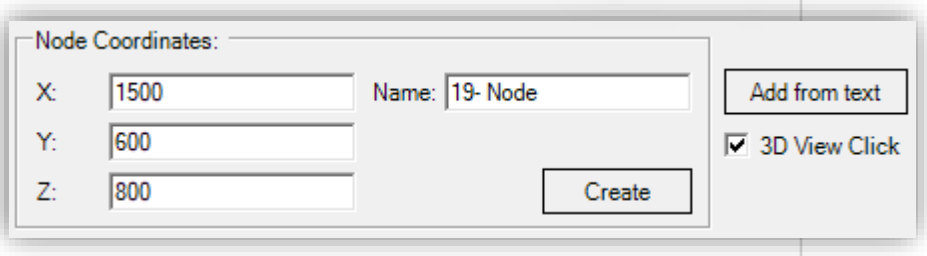
A Deformable Barrier is defined on a single Node present in a VCS Solution.
In consequence, in order to create a deformable barrier, the user needs to create a Node first.
Afterwards the user should select the "Create Barrier" tool from the VCS main toolbar and select one of the given barrier types.

Create

Step 1: "Create Node" tool

The „Create Node" tool can be found in the VCS main toolbar. The basic Node creation procedure is given below:





1) Select the "Create Node" tool from VCS main toolbar.

2) In the "Tool Settings" window you can set the coordinates of newly created Nodes.

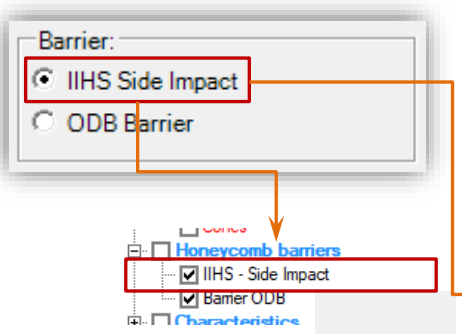
3) You can also easily name each Node you have created.
The name of the new Node will be automatically added to the list of nodes in the Solution Explorer window.

Barrier

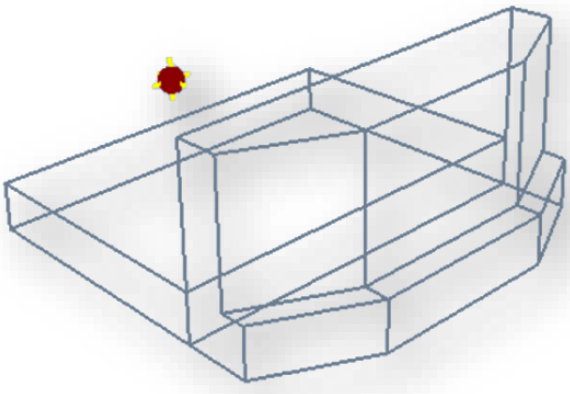
Step 2: "Create Barrier" tool

Visual Crash Studio offers two types of pre-defined deformable barriers.
A Deformable Barrier is defined on a single Node after selecting the "Create Barrier" tool from the VCS main toolbar and clicking on the required Node in the Solution's 3D view.

1) IIHS Side Impact barrier



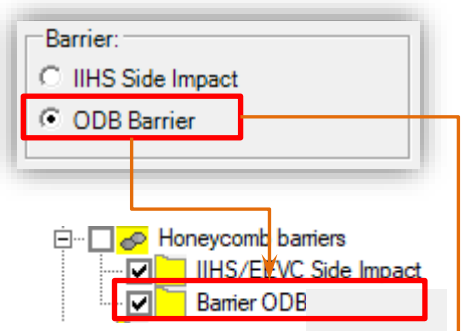
In the Tool Settings window the user can specify what type of barrier is to be created.
First deformable barrier type available in VCS is the IIHS Side Impact barrier.



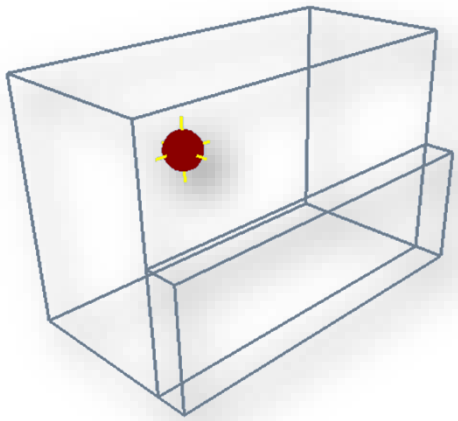
New barrier is automatically displyed in the 3D view and added to the Solution Explorer window

"Create Deformable Barrier" tool

2) ODB barrier



In the Tool Settings window the user can specify what type of barrier is to be created.
Second deformable barrier type available in VCS is the ODB barrier.



New barrier is automatically displyed in the 3D view and added to the Solution Explorer window

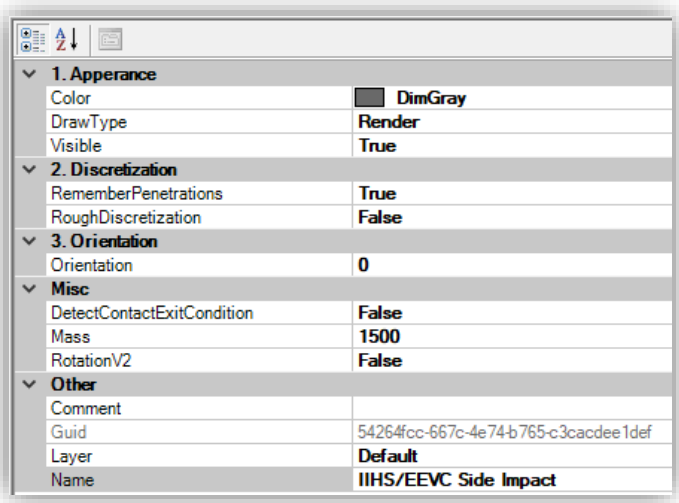
DEFORMABLE BARRIER - Properties

In the deformable barrier's Properties window you can set the details of object's appearance (color, visibility).

In the "Orientation" section you can define barriers position in the 3D space by indicating the degrees of rotation around the Z axis.

Please note that the mass of a deformable barrier is set by default and cannot be changed by the user.

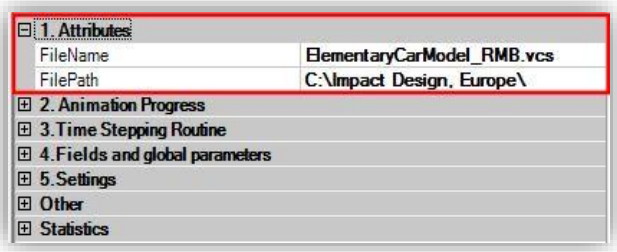
Velocity of the barriers should by defined in the "Kinematic Constraints" section of the Nodes Properties window.

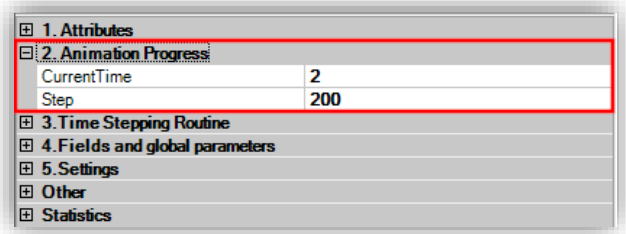


1. Apperance	
Color	DimGray
DrawType	Render
Visible	True
2. Discretization	
RememberPenetrations	True
RoughDiscretization	False
3. Orientation	
Orientation	0
Misc	
DetectContactExitCondition	False
Mass	1500
RotationV2	False
Other	
Comment	
Guid	54264fcc-667c-4e74-b765-c3cacdee1def
Layer	Default
Name	IIHS/EEVC Side Impact

SOLUTION SETTINGS

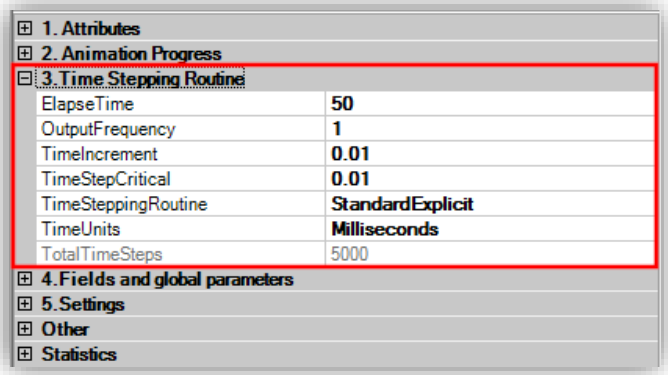
Detailed Solution parameters are displayed after selecting the top-most object in Solution Explorer tree. The first property section describes ***Attributes*** *of the opened file, i.e., file name and file location on the hard disc.*





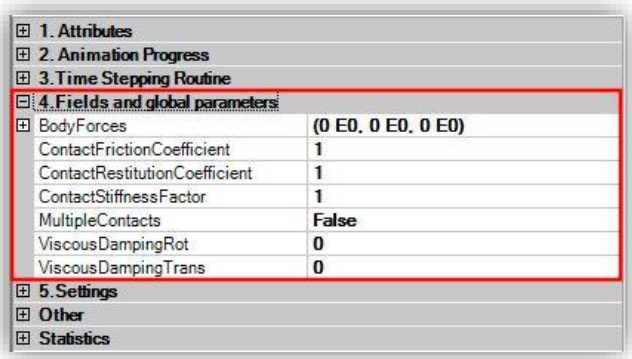
Second section - Animation Progress corresponds to animation menu placed on the top of interface window. Here the number of time step congruent with current time of the simulation is displayed.

Time Stepping Routine section is directly related with simulation parameters. It contains listed elements:



- ***Elapse Time value*** *which determines the time of the event simulation.*
- ***Output Frequency*** *- responsible for limitation of the result data stored in computer's memory. Default value equals 1. That means 1 save of the results will be done at each iteration step.*
- ***Time Increment*** *– elapse time divided by time increment value gives Total Time Steps.*
- ***Time Stepping Routine*** *– Standard Explicit time stepping routine is implemented in VCS.*
- ***Critical time Step for the explicit routine*** *– mass and moments of inertia are added where necessary to get stable results.*
- ***Time Units*** *– as default the milliseconds are set. Other selection contains seconds and microseconds.*

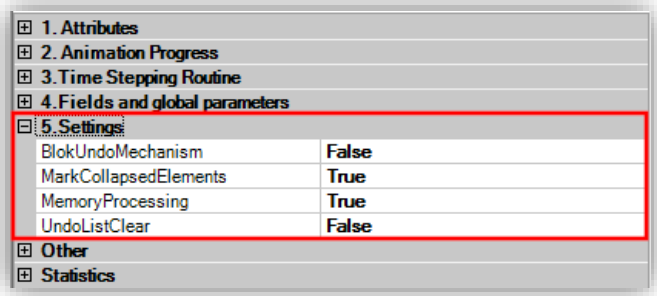
The Fields and global parameters *section defines body forces and general settings for viscous forces and contact events.*



- ***Contact restitution coefficient***
  Value 0 perfectly plastic
  Value 1 purely elastic
- ***Multiple Contact*** *switches on all possible contact envelopes and checks the contact between each element in 3D structure.*

SOLUTION SETTINGS

*General **Settings** section of the Solution's Properties window regards the usage of interface operations.*



- ***Block Undo/Redo Mechanism** reduces the usage of computers memory.*

- ***Mark Collapse Elements** – the user can choose whether damaged, collapsed elements will be marked*

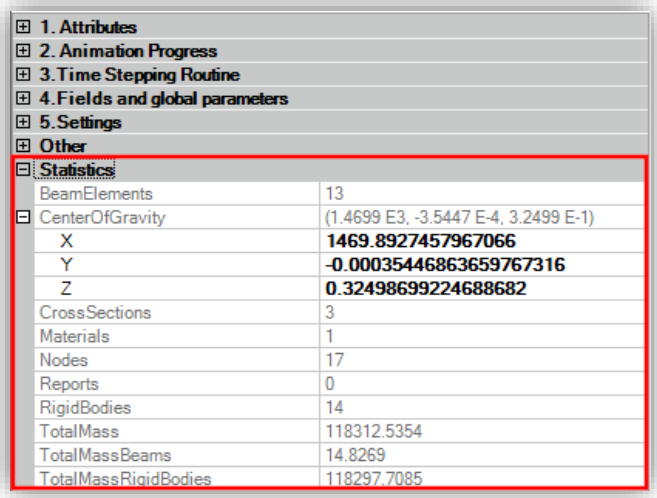
- ***Memory Processing** – When the Memory Processing flag is set to 'True' the results of the simulation will be stored in the RAM memory. This is a fast option recommended for small models).*
- *When the Memory Processing flag is set to 'False' the results of the simulation will be stored in a separate folder on the computers HDD in the location of the currently opened Solution.*

- ***Undo List Clear** – If the Undo List Clear fag is set to 'True' the Undo list will be cleared prior each calculation / save process (this grants free memory space for runtime data)*

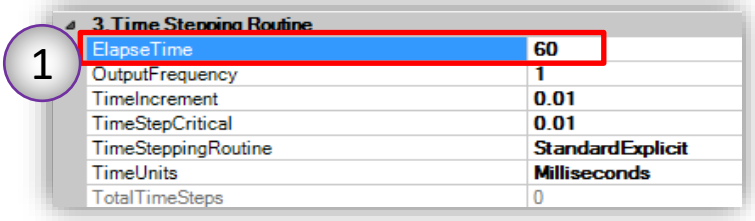
*The **Statistics** section of the Solutions Properties window is especially useful at the final stage of model checking.*

This section gathers the information about the number if elements of the model and the mass of the model.

Moreover, you will find the calculations of Centre of Gravity



TIME STEPPING ROUTINE – STEP BY STEP



STEP 1

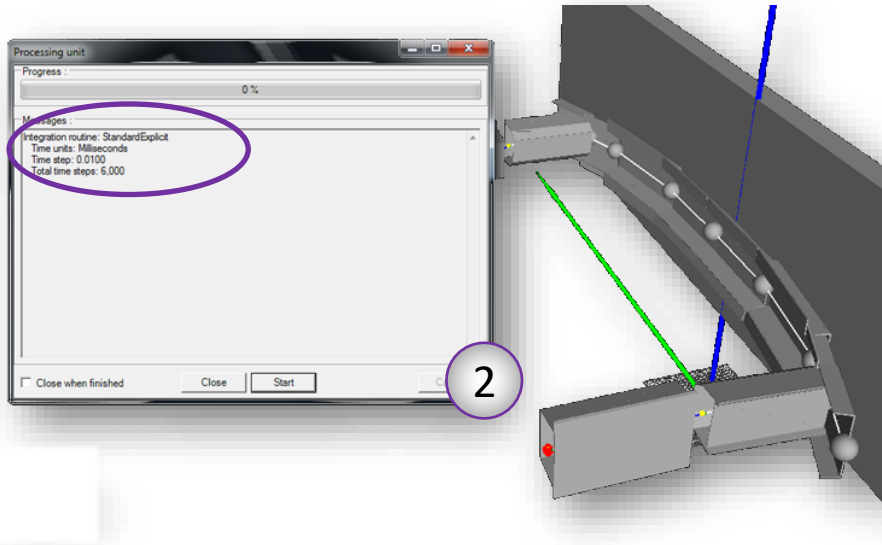
*In the Solutions Properties window set the **Elapse Time** of the simulation.*
In the presented example Elapse Time has been set to 60 milliseconds.

STEP 2

Click on the "Calculate" icon in the VCS main toolbar and initialize the calculation process.

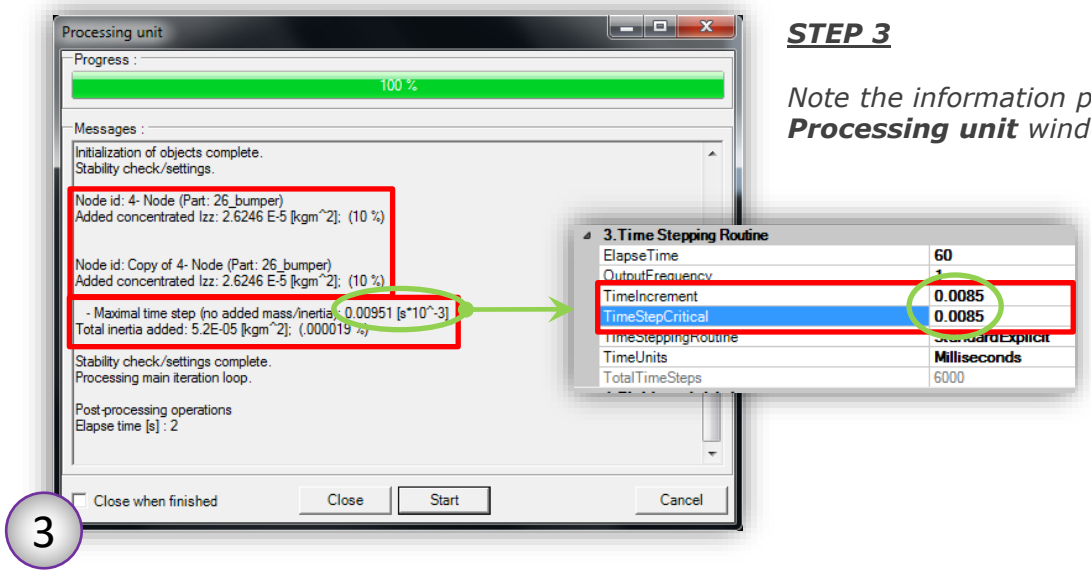
*The **Processing unit** window, which will appear on the screen, includes basic information about the current time stepping routine.*

Click on "Start" button – the model check and calculation process will be conducted.



STEP 3

*Note the information provided in the **Processing unit** window.*



The processor will give information about added concentrated mass – added mass indicates the necessity of time step modification.
Processing unit provides the maximum value of time step.

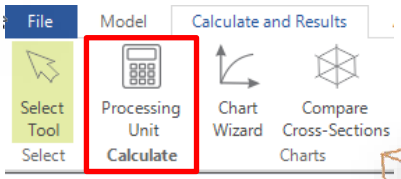
*Basing on those information **Time Step Critical** and **Time Increment** should be defined in the **Solutions Properties window** – the value should be smaller than the one provided in the Processing unit window*

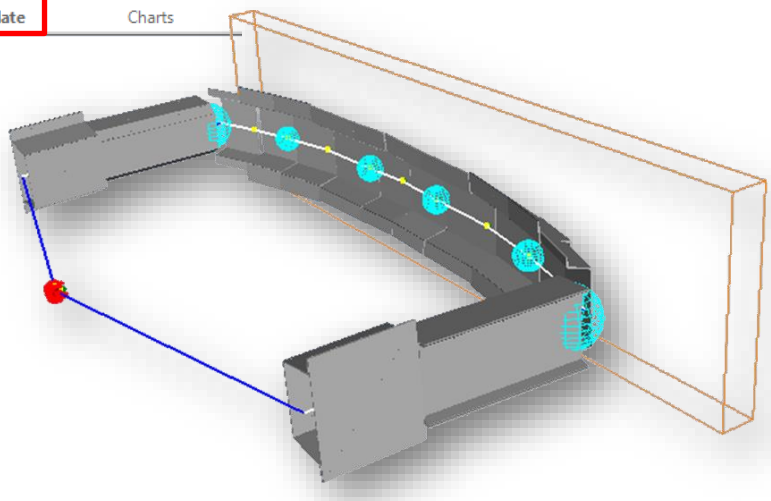
CALCULATE THE SOLUTION

In order to start the calculation process of the Solution, simply click on the „Processing Unit" icon on the main toolbar ("Calculate" section).

You can also right click on the Solution folder in the Solution Explorer window and select the calculate option.

A 'Processing unit' window will be visible on the screen. After clicking on the 'Start' button the calculation process will begin.

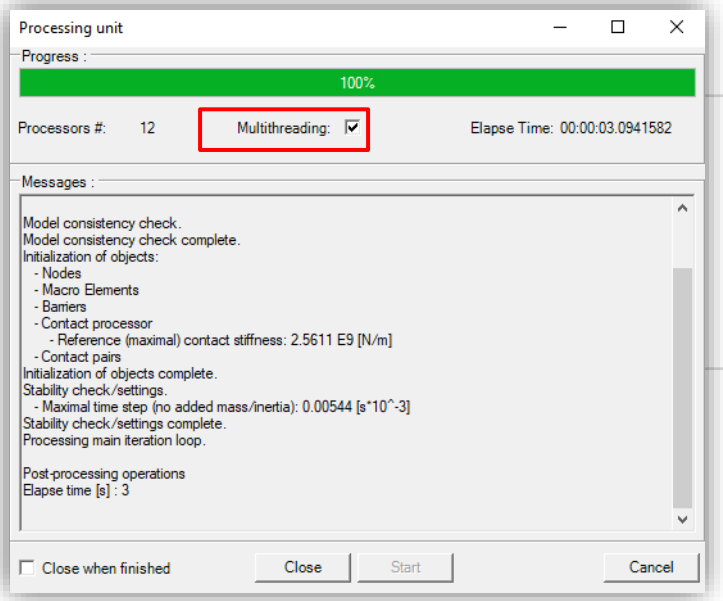




You can monitor the progress of the calculations in the 'Progress' field of the 'Processing unit' window. In the 'Messages' section of the window you can find information about the Integration routine of the simulation, time units, time step and total time steps.
Before the calculation process starts the model consistency check will be automatically conducted.

CALCULATIONS - MULTITHREAD

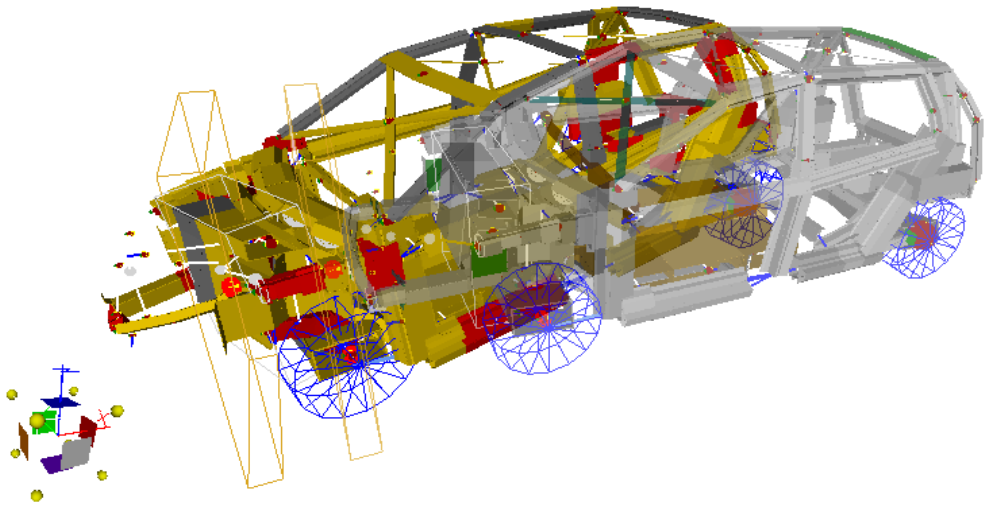
In VCS 6.0 multithread calculation routine is now possible.
This improvement is especially useful in case of large VCS models, in which case the calculation time can be reduced by approximately 30%!



*In order to enable the multithread calculations click on the **"Multithreading"** check box in the "Processing unit (available after clicking on the "Calculate" icon in VCS main toolbar)*

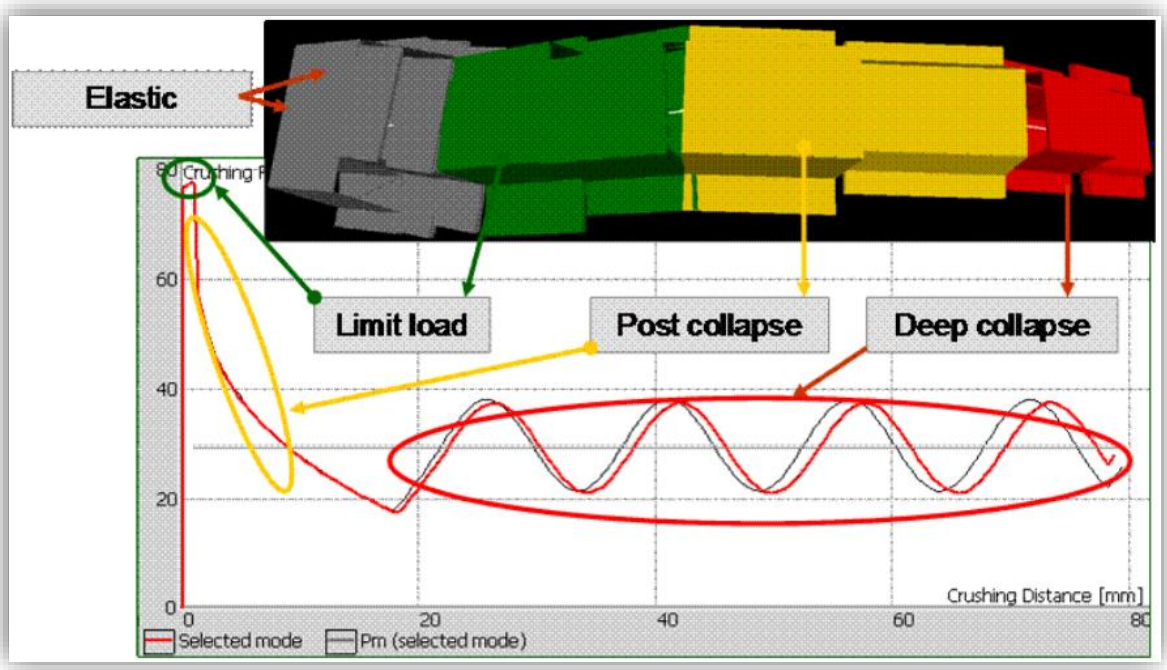
Animation

After the calculations are completed, in the 'Processing unit' window you will find information about the elapse time of the simulation.
After completing the calculation process you can view the animation of the model's load case. In order to start the animation click on the 'Play' icon on the main toolbar.

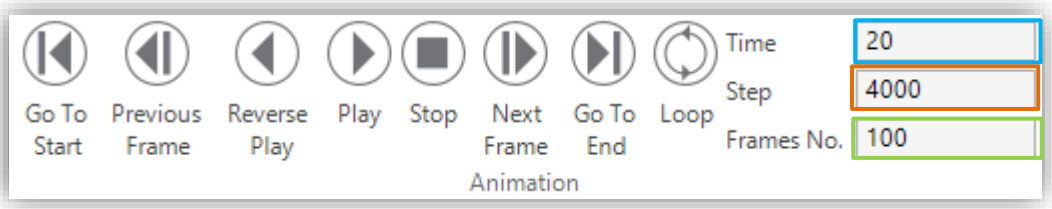


During the animation you can observe the movement and deformation of the model. The color scheme illustrated in the picture below is implemented in order to simplify interpretation of simulation results:

- ***Dark gray**: macro elements which remain in elastic range*
- ***Dark green**: elements that have reached limit load carrying capacity (under a combined loading).*
- ***Dark Yellow** (golden): elements in the post collapse range*
- ***Red**: elements in deep collapse region,*



ANIMATION TOOLBAR



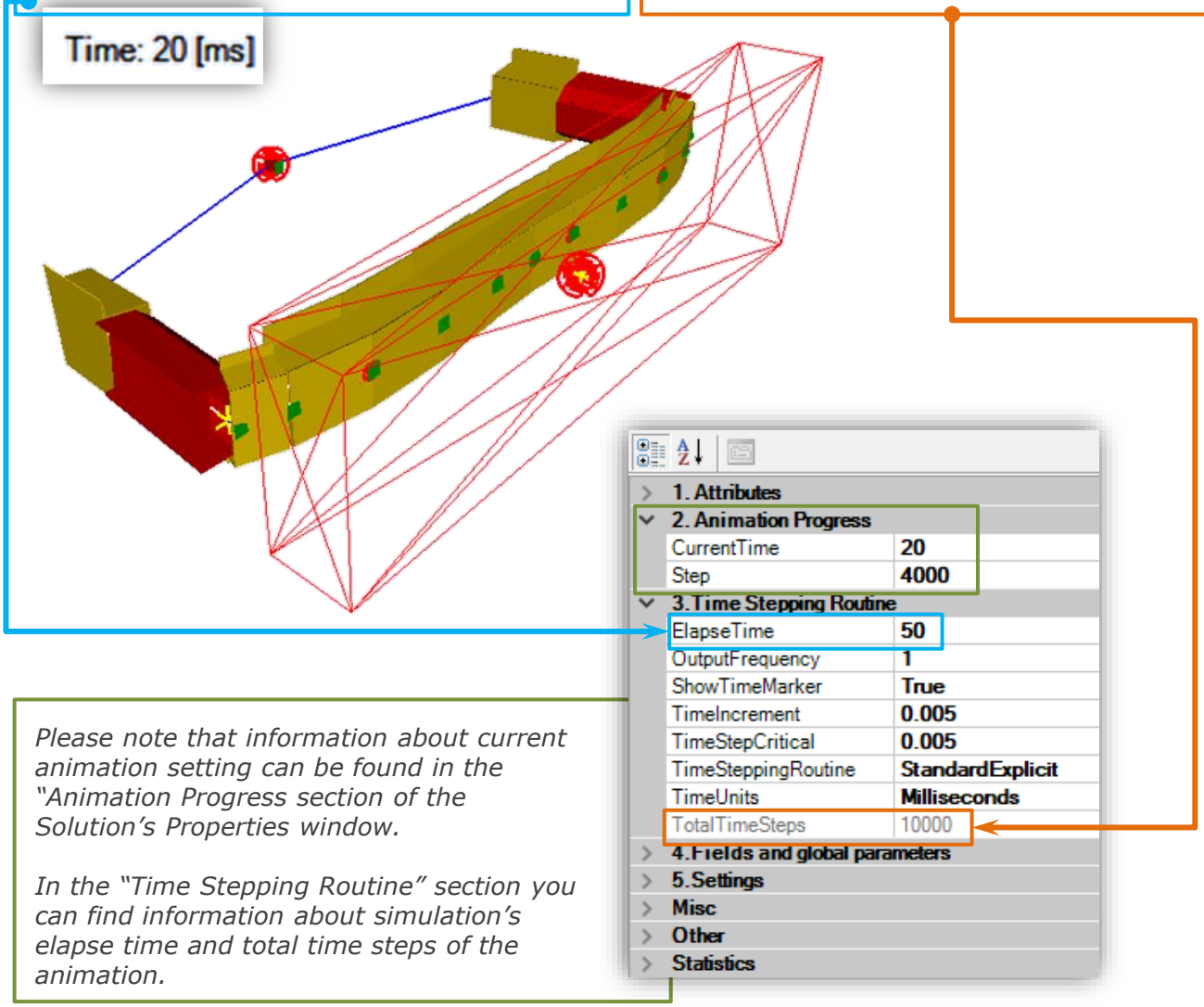
NUMBER OF FRAMES IN SIMULATION

CURRENT SIMULATION TIME

Enter a requested time of the animation in order to see the structure's deformation stage in this particular moment.
Time is set in [ms].

SIMULATION STEP

Enter a requested simulation step in order to see the structure's deformation stage in this particular moment.



Please note that information about current animation setting can be found in the "Animation Progress section of the Solution's Properties window.

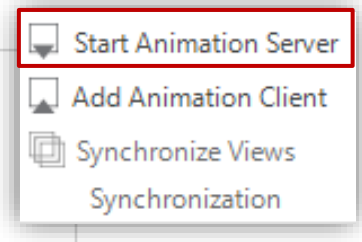
In the "Time Stepping Routine" section you can find information about simulation's elapse time and total time steps of the animation.

Synchronization Functionality

Synchronization Functionality *provides the possibility to view and compare several VCS files simultaneously.*

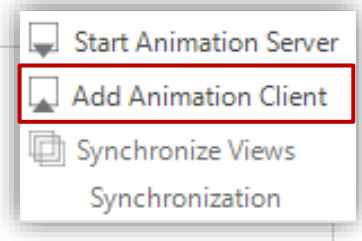
1

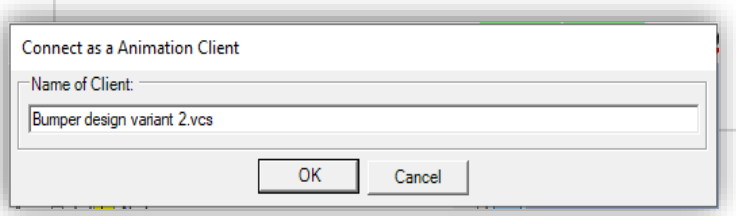
In the first VCS file click on the "Start Animation Server" icon in the VCS main toolbar.
This file will be set as the "Server" file which means that it will be the master file managing the animation of all other "Client VCS files



2

In the second VCS file click on the "Star Animation Client" icon.
In the "Connect as Animation Client" window define the name of client file and click "OK"



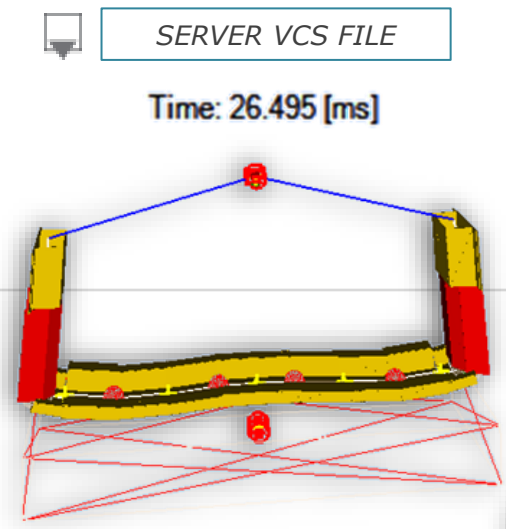


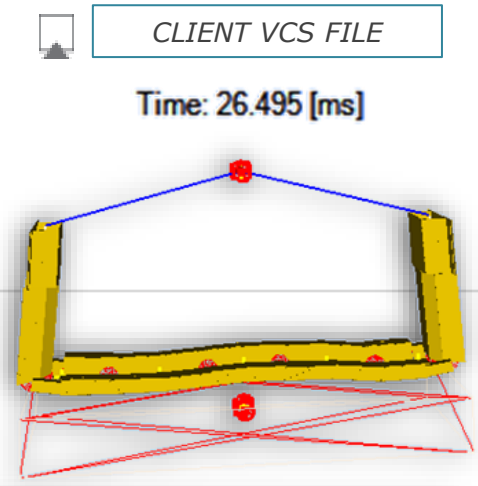
The animation of the client file will be synchronized with the server file.
As presented in the pictures below, whenever the animation is started or stopped the animation in the client file will act accordingly.
Simulation time will always be exactly the same as in the server file

SERVER VCS FILE

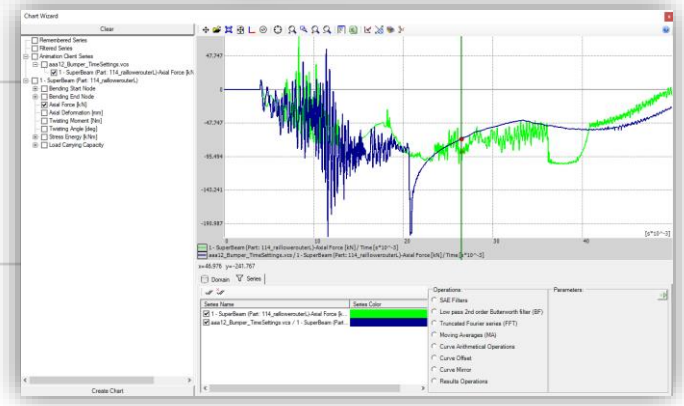
3

CLIENT VCS FILE

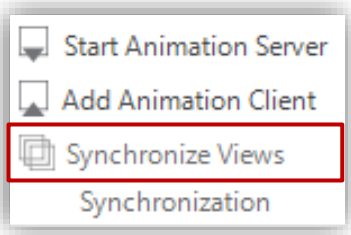




Additionally, in the Chart Wizard window you will be able to compare results of the server and client load case

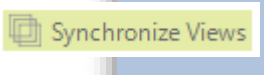


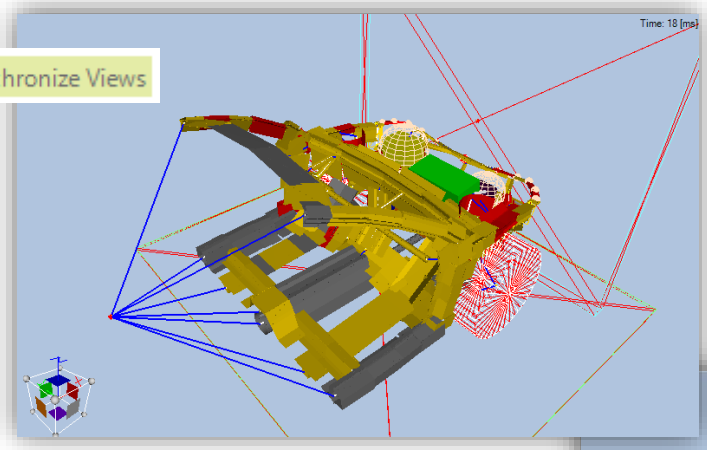
Synchronization Functionality



Synchronize Views *functionality.*

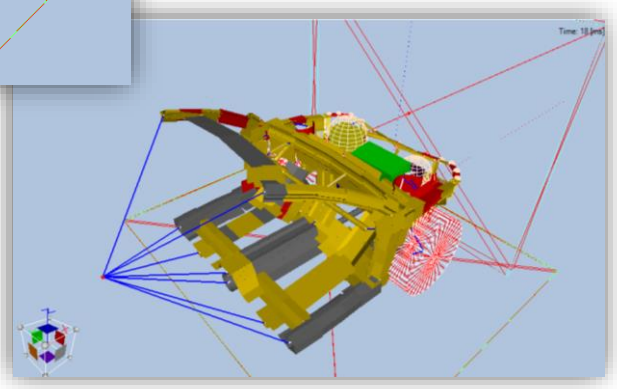
After clicking on the "Synchronize Views" icon any rotation, shift or zoom operation will be automatically repeated in the "client" VCS file





SERVER VCS FILE

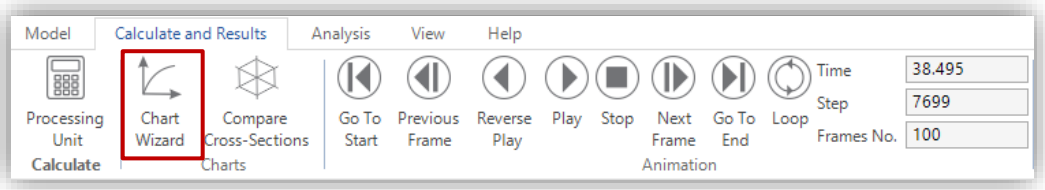
Please notice that synchronized view changes can be made only in the "server" VCS file.



CLIENT VCS FILE

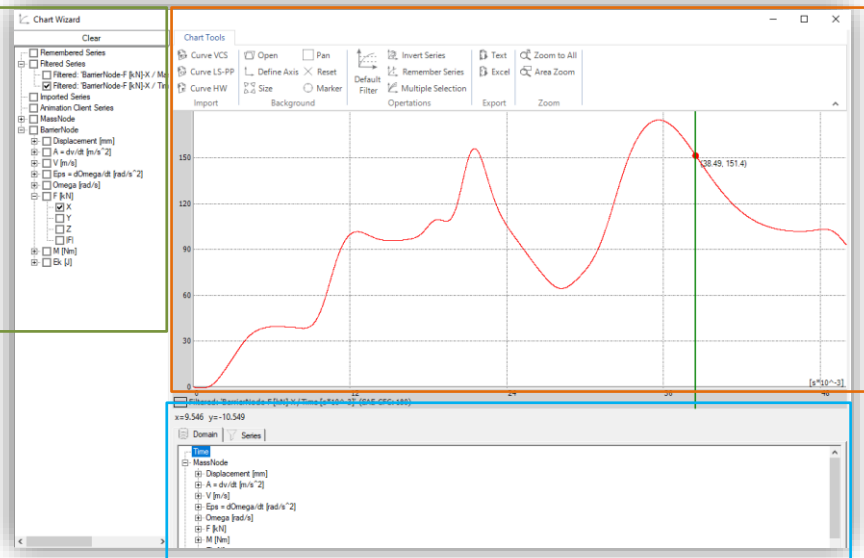
Chart Wizard

Chart Wizard is used to create various results graphs.
The icon "Chart Wizard" is located on the main toolbar of the "Calculate and Results" section



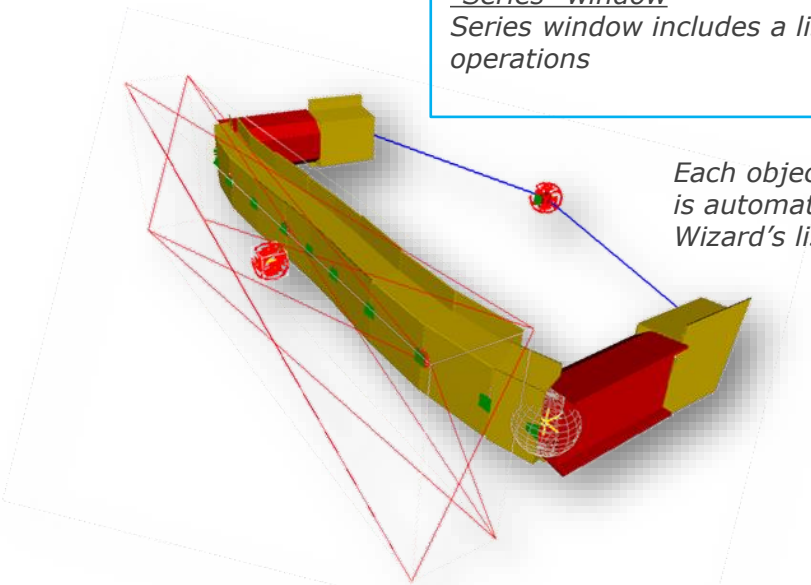
After selecting the Chart Wizard icon a separate window will be visible on the screen:

Selection window for dependent variable (ordinate). The default independent variable (X axis) is time.



The main window presents the function chart. Green time marker line illustrates the moment of the load case simulation.

"Domain" window
List window at the bottom of Chart Wizard enables selection of function domain.
"Series" window
Series window includes a list of available curve operations



Each object of a VCS model selected in the 3D view is automatically added to the list in the Chart Wizard's list window on the left-hand side.

(*) For more information about the CHCAR WIZARD functionality please see the "VCS 8.0 – Chart Wizard" manual.