Oasys Primer

Column Blast Tool – Ver 3.0

User Manual

Issue 1 | April 6, 2021

Job number 076023-37

Arup North America Ltd
560 Mission Street
Suite 700
San Francisco 94105
United States of America
www.arup.com

ARUP

Contents

Page

1 Introduction 1

2 Model Creation 2

2.1 Main GUI 3
2.2 General Column Details 4
2.3 Steel Sections 5
2.4 Concrete Sections 6
2.5 Material Properties 8
2.6 Charge Details 9
2.7 Control Parameters 10

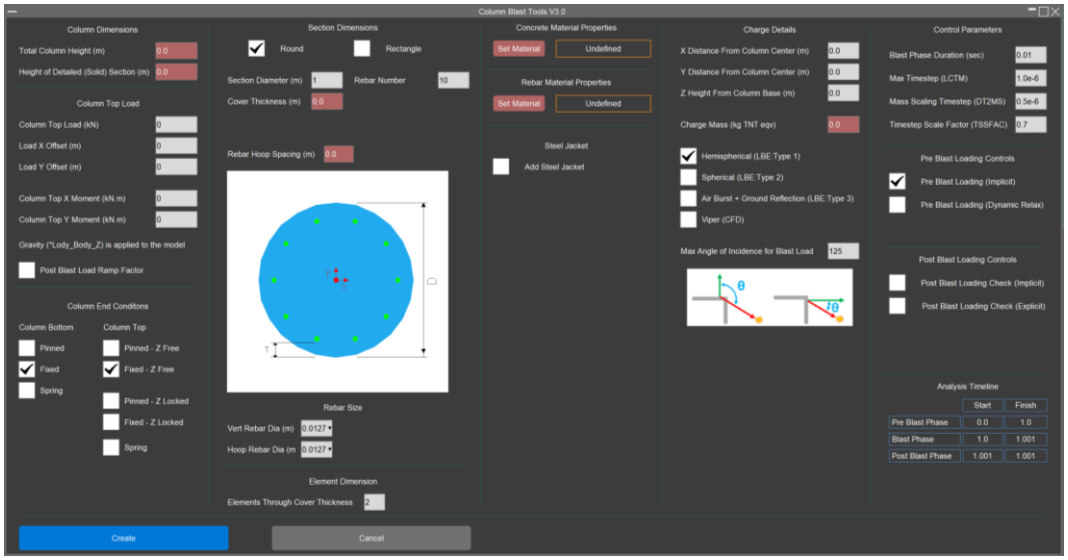
3 Model Setup 11

3.1 Rebar 12
3.2 Boundary Conditions 13
3.3 Gravity and Column Top Load 14
3.4 Blast Loading 14
3.5 Materials and Element Type 15
3.6 Contact 16
3.7 Ground 16

4 Additional Files 17

1 Introduction

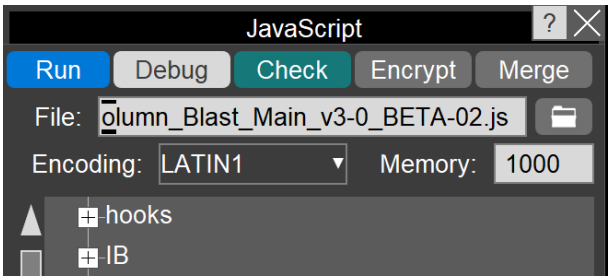
This Oasys Primer JavaScript allow the user to quickly setup a column blast analysis for a range of steel and reinforced concrete section types.



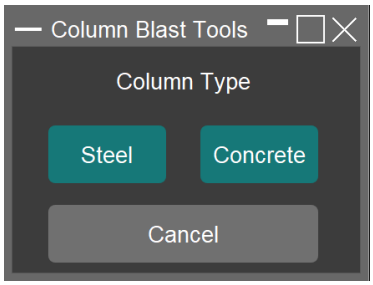
Along with the main JavaScript (Column_Blast_v3-0.js) several JS modules, macros data, and image files also required. These files are located in various folders. The full list of these additional files is given Section 4.

2 Model Creation

It is recommended when using this Oasys Primer JavaScript to set the memory to 1000 due to the number of elements being processed.



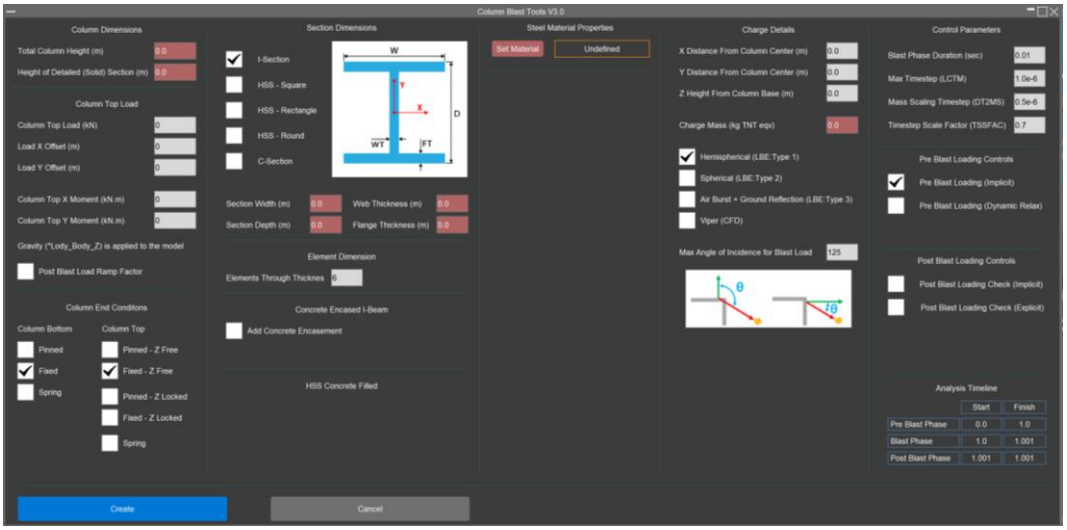
The user first selects whether they are creating a steel or reinforced concrete column.

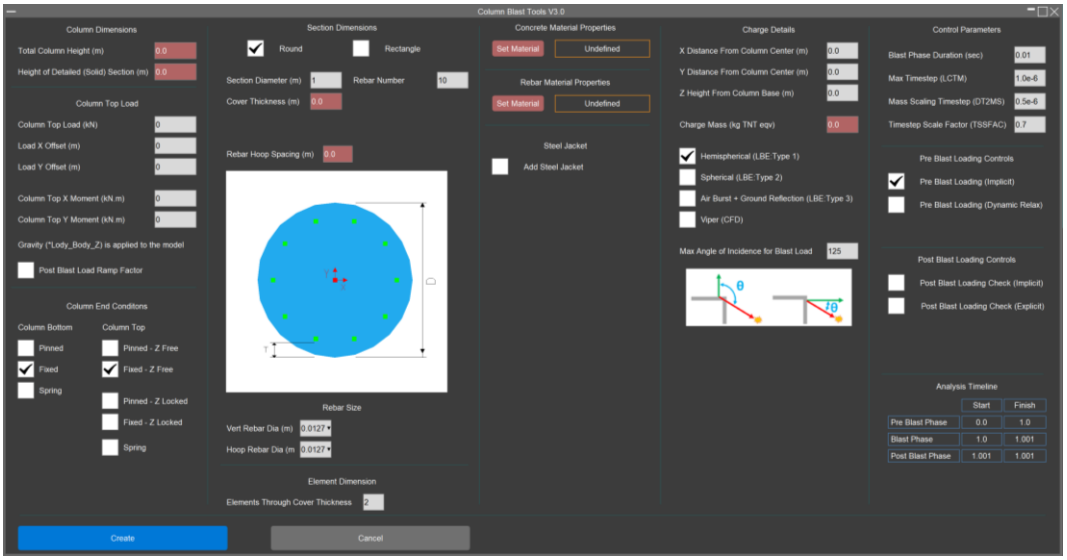


From there the user is taken to either the Steel or Reinforced Concrete GUI.

2.1 Main GUI

The main GUIs feature a series of sections that need to be completed by the user.





- The red entries are required to be filled out.
- The white/blue entries have suggested values but these need to be reviewed by the user for applicability for their analysis.

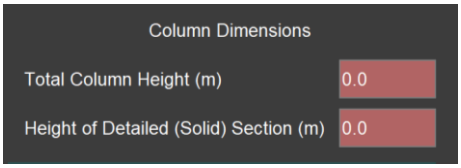
These sections include;

- Column Dimensions
- Column Loading
- Section Properties
- Material Properties
- Charge Details
- Control Parameters

2.2 General Column Details

Column Dimensions

The user enters the total height of the column and the height of the section of the column to be modelled in solid elements.



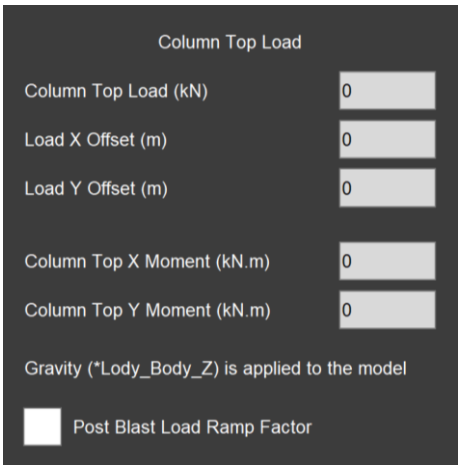
Column Top Load

The user specifies the loads and moments to applied to the top of the column.

The top load will be applied in the negative z direction.

An offset can be applied to the top vertical load.

The user can also specify a ramp factor that will be applied to the loads and moments in the post blast phase.



Column End Conditions

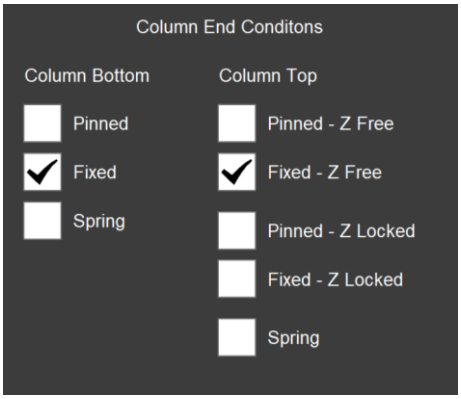
The user selects the end conditions (Pinned, Fixed, Spring) for the top and bottom of the column.

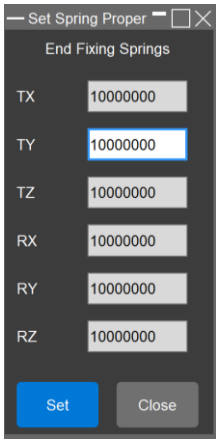
If the Spring option is selected then the user has to enter the six spring stiffness (3 translation, 3 rotational) by clicking on the ‘Set Spring’ button.

The spring stiffness can be pre-setup by editing the;

- *bottom_spring.csv*
- *top_spring.csv*

files in the *Spring_Properties* folder



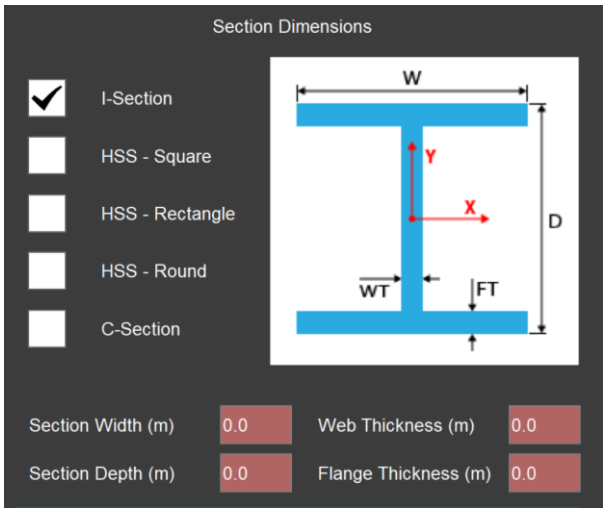


2.3 Steel Sections

Section Dimensions

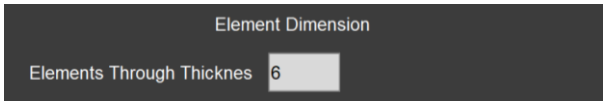
The user selects the section type and enters the relevant dimensions.

- I-Section
- HSS – Square
- HSS – Rectangle
- HSS – Round
- C-Section



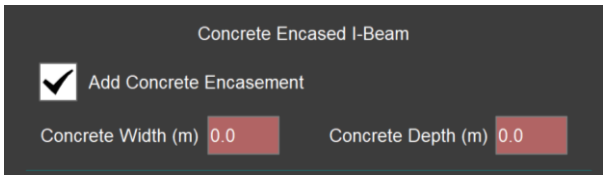
Element Dimensions

The user selects the number of solid elements to be used through the thickness of the steel plates.



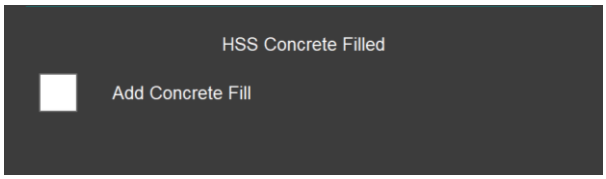
Encased I-Section

There is the option for an I-Beam to be modeled encased in concrete.



Filled HSS Sections

There is the option to for a HSS section to be modeled filled with concrete.



2.4 Concrete Sections

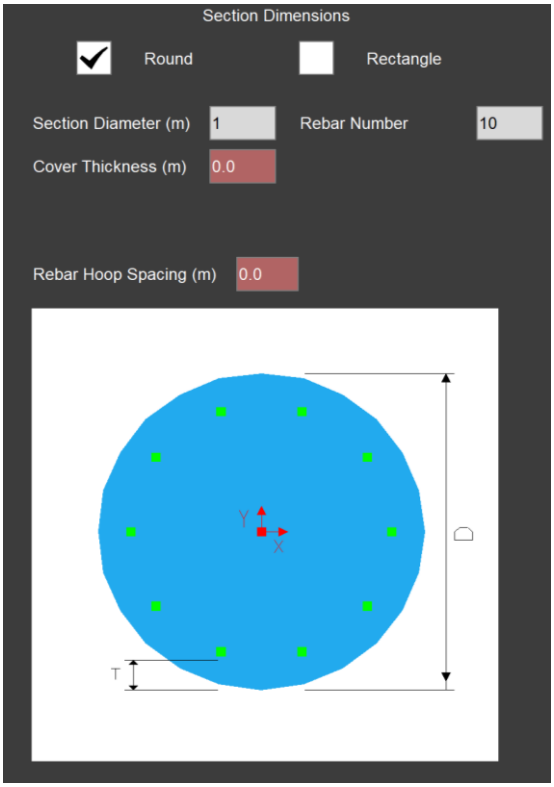
Section Dimensions

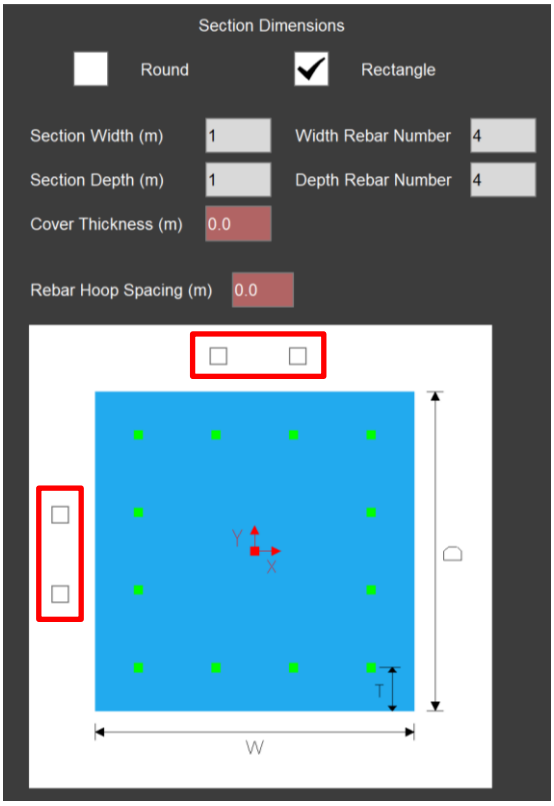
The user selects the section type and enters the relevant dimensions.

- Round
- Rectangle

For a rectangular section the user can use the check boxes on the cross-section diagram to select the location of the rebar stirrups.

The diagram showing the cross section will dynamically update as the user changes the number of rebar and selects which stirrups to include.

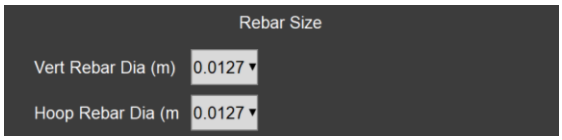


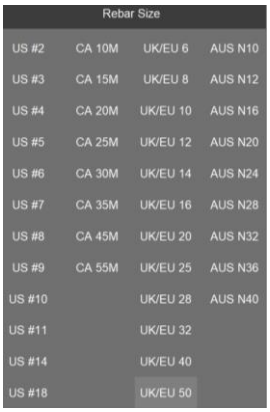


Rebar Setup

The user selects rebar diameter for the vertical, hoop and stirrups.

There is a drop-down menu from which the user can select standard US, Canadian, UK/EU and Australian rebar sizes.





Element Dimensions

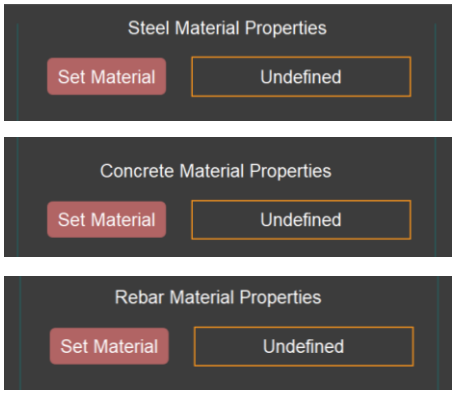
The user selects the number of solid elements to be used through the thickness of the cover concrete.

Steel Jacket

The user can select to add a steel jacket to the outside of the concrete column.

2.5 Material Properties

The user can set material properties by clicking on the ‘Set Material’ button.

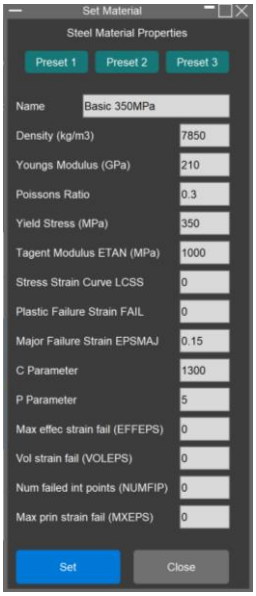


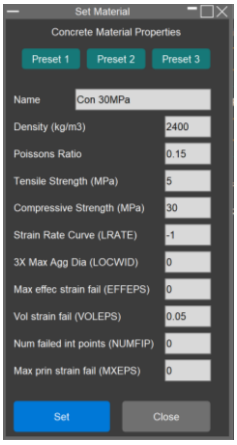
This will bring up a window for the relevant material type (Steel, Concrete, Rebar).

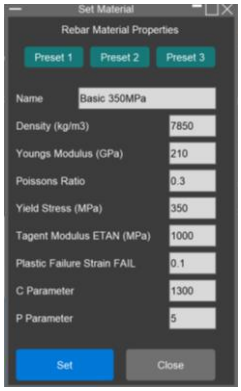
The current LS-DYNA material models used are;

- Steel: Mat_123
- Concrete: Mat_72R3
- Rebar: Mat_24

There are three preset material that the user can select from or the user can edit the values to create their own material.







To use a stress strain curve (LCSS) for a steel material, the user needs to pre-create the load curve/load table and save as a keyword file. The user will be prompted to select this keyword file when they click the ‘Set’ button. The id number entered in the LCSS text box needs to match the id number in the keyword file.

There are example load curve and load table keyword files in the ‘Material_Properties’ folder.

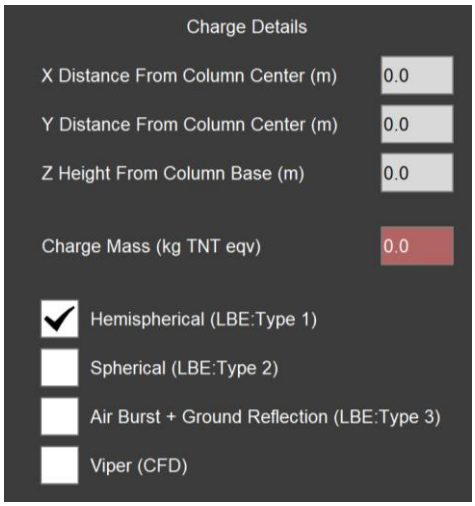
The preset materials can be changed by editing the;

- *steel_material.csv*
- *concrete_material.csv*
- *rebar_material.csv*

files in the *Material_Properties* folder.

2.6 Charge Details

Charge Details



The user enters the x, y and z coordinates of the charge relative to the center of the column

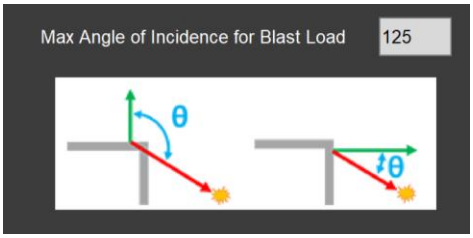
The user enters the charge mass in kilogram TNT equivalent.

The user can select from three types of Load_Blast_Enhanced charge.

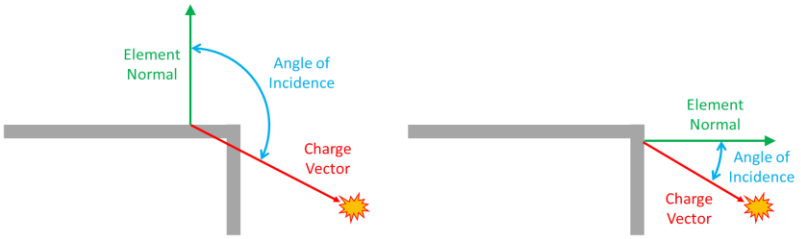
- Hemispherical
- Spherical
- Air burst charge.

The user can also select a Viper (CFD) charge. In this case a second model is created that can be imported into the Viper CFD analysis to setup the column geometry. The result from the CFD analysis will then need to be imported into the main model.

The user enters the max angle of incidence for which the blast load should be applied to the structure



The angle of incidence is calculated by comparing the angle between the element normal vector and the vector between the element center and the charge location.

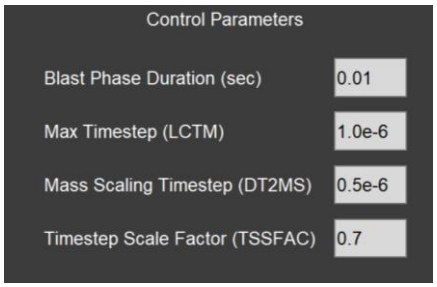


A max angle of incidence greater than 90 deg may be required if the blast load needs to be applied to the sides faces as well as the front face of the section.

*Set_Segment id 200 (id 300 for I-section) will contain the segments used for the blast loading.

2.7 Control Parameters

The user can set various basic control parameters for the model.



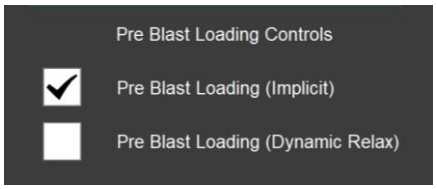
Pre-blast loading

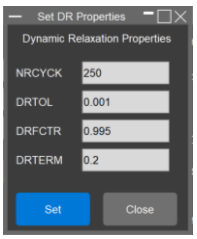
The user can select to include a pre-blast loading phase where the gravity and top loads will be applied to the model.

The user can select either an Implicit or Dynamic Relaxation (DR) analysis for this phase.

If DR is chosen, there is a pop-up window in which the DR properties can be set.

The pre-set DR properties can be edited in the *dr_prop.csv* file in the *Dynamic_Relaxation_Properties* folder.





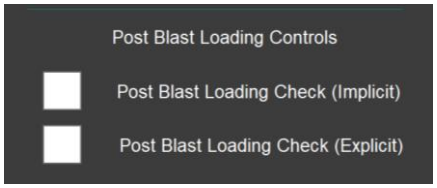
Post-blast loading

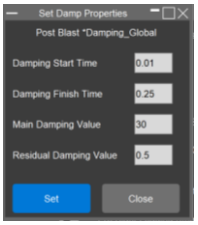
The user can select to include a post-blast loading phase where the gravity and top loads will held constant or ramped up.

The user can select either an Implicit or Explicit analysis for this phase.

If Explicit is chosen, then the user has to set the duration for this phase and post blast damping properties.

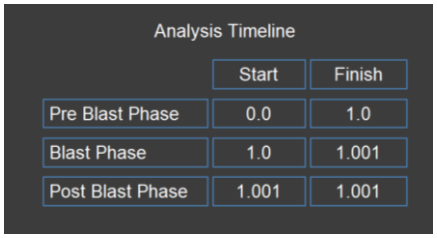
The pre-set damping properties can be edited in the *damping_global.csv* file in the *Damp_Properties* folder.





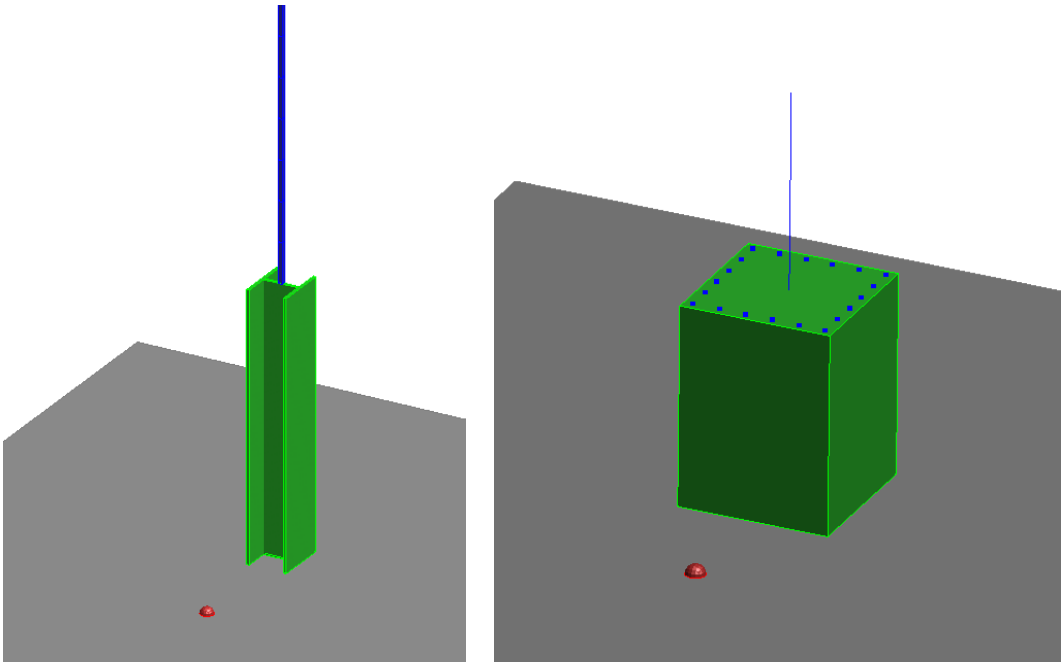
Analysis Timeline

An overall analysis timeline is also shown for reference.



3 Model Setup

The column is modeled using solid elements for the lower section where the blast load is applied and beam elements for the upper section of the column.



- An initial pre-load phase can be included to apply gravity and top load (if any) to the column. This can be either an implicit (static) or dynamic relaxation phase.
- The analysis then switches to an explicit (dynamic) analysis for the blast loading.
- A final post-load phase can be included to determine if the post-blast column still has the capacity to carry the gravity and top load. This can be either an implicit (static) or explicit (with damping) phase. A factor can also be applied to the loads to ramp them up during this phase

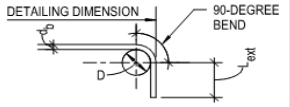
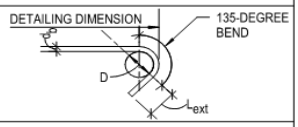
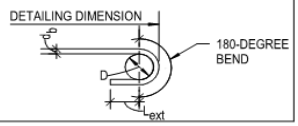
3.1 Rebar

For a concrete column the rebar are modelled using beam elements. The connection between the rebar and the concrete is modelled by having the beam and solid elements meshed together (i.e. sharing nodes).

The interaction between the different rebar is modelled using a contact definition that accounts for the thickness of the beam elements.

The hook lengths for the stirrup rebars are taken from our standard US structural details which are based on the AISC code. These are shown in the following table.

STANDARD HOOK GEOMETRY FOR STIRRUPS, TIES, HOOPS

TYPE OF STANDARD HOOK	BAR SIZE	MINIMUM INSIDE BEND DIAMETER D, in.	STRAIGHT EXTENSION L, in.	TYPE OF STANDARD HOOK
90-DEGREE HOOK	No. 3 THROUGH No. 5	$4d_b$	GREATER OF $6d_b$ and 3 in.	
	No. 6 THROUGH No. 8	$6d_b$	$12d_b$	
135-DEGREE HOOK	No. 3 THROUGH No. 5	$4d_b$	GREATER OF $6d_b$ and 3 in.	
	No. 6 THROUGH No. 8	$6d_b$		
180-DEGREE HOOK	No. 3 THROUGH No. 5	$4d_b$	GREATER OF $4d_b$ and 2.5 in.	
	No. 6 THROUGH No. 8	$6d_b$		

3.2 Boundary Conditions

The user can select either pinned, fixed or spring boundary conditions for the top and bottom of the column.

Top Restraint

For the top restraint a *Boundary_SPC is used on the top node of the last beam element.

For pinned (Z free) the restraint at the top will be;
- X, Y translation
- Z rotation

For pinned (Z fixed) the restraint at the top will be;
- X, Y, Z translation
- Z rotation

For fixed (Z free) the restraint at the top will be;
- X, Y translation
- X, Y, Z rotation

For fixed (Z fixed) the restraint at the top will be;
- X, Y, Z translation
- X, Y, Z rotation

For spring the user defines the spring stiffness for each of the six degrees of freedom.
- X, Y, Z translation
- X, Y, Z rotation

Bottom Restraint

For the bottom restraint a *Constrained_Nodal_Rigid_Body with the SPC option is used for the solid element nodes at the bottom of the column.

For pinned the restraint at the bottom will be;
- X, Y, Z translation
- Z rotation

For fixed the restraint at the bottom will be;
- X, Y, Z translation
- X, Y, Z rotation

For spring the user defines the spring stiffness for each of the six degrees of freedom.

- X, Y, Z translation
- X, Y, Z rotation

3.3 Gravity and Column Top Load

Gravity is applied to the model using the *Load_Body_Z card, with g =9.81 N/m^2

Column load and moments are applied to the top node of the last beam element using a *Load_Node card based on the value entered by the user.

- The top load acts in the negative Z direction.
- If an offset is defined for the top load. The resulting moment are calculated and applied to the top node.

3.4 Blast Loading

The blast loading is applied using the *Load_Blast_Enhanced card and a *Set_Segment. The user selects;

- The charge location (x, y, z cords).
- The charge mass in kilogram TNT equivalent.
- Either a spherical, hemispherical, or airburst charge.

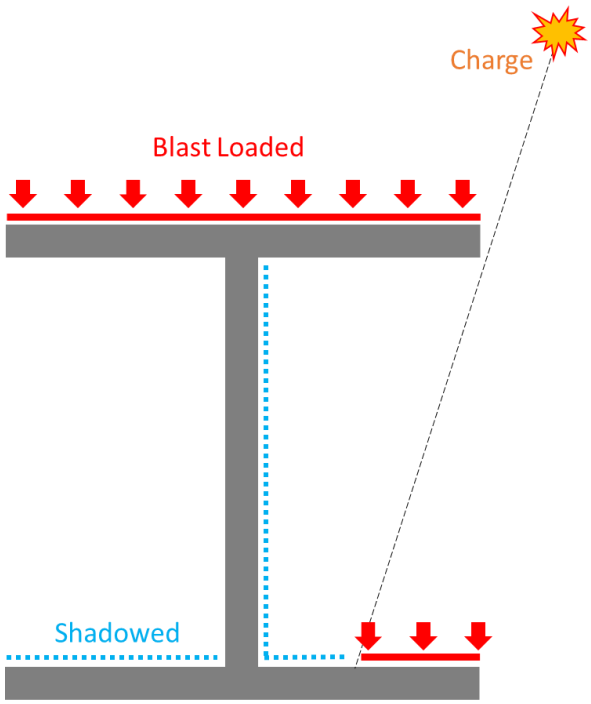
A Viper (CFD) loading option is also available. If the user selected this option a second model is created with the blast segment set and column outer surface solid elements. This model can be used for importing into Viper for setting up the model geometry and data output locations.

A shell element sphere/hemisphere is created at the location of the charge for visualization.

A series of five *Rigidwall cards are used to surround the model and prevent any eroded element nodes from flying off and causing error termination.

The JavaScript calculates which segments to apply the load to based on the charge location and the max angle of incidence entered by the user.

For I-Section and C-Section profiles the JavaScript includes a routine that calculates which segments are in the blast shadow of other surfaces and does not apply the blast load to these segments.



3.5 Materials and Element Type

3.5.1 Steel

For the solid elements.

- The *Mat_Modified_Piecewise_Linear_Plasticity (Mat_123) material is used.
- The strain rate effects are defined by the user. The Cowper Symonds rule (C and P parameters) is used to include the strain rate effects.
- The type 2 fully integrated S/R solid element formulation is used

For the beam elements.

- The *Mat_Plastic_Kinematic (Mat_003) material is used.
- There is no failure defined
- There are no strain rate effects.
- The type 1 Hughes-Liu beam element formulation is used

3.5.2 Concrete

For the solid elements.

- The *Mat_Concrete_Damage_REL3 (Mat_72R3) material is used.
- The type 1 solid element formulation is used

For the beam elements.

- The *Mat_Elastic (Mat_001) material is used.
- There are no strain rate effects.
- The type 1 Hughes-Liu beam element formulation is used

3.5.3 Rebar

For the rebar beam elements.

- The *Mat_Piecewise_Linear_Plasticity (Mat_24) material is used.
- The strain rate effects are defined by the user. The Cowper Symonds rule (C and P parameters) is used to include the strain rate effects.
- The type 1 Hughes-Liu beam element formulation is used

3.6 Contact

A *Contact_Eroding_Single_Surface is applied to the solid element part to prevent elements from passing through each other as the column deforms.

A *Contact_Automatic_General is applied to the beam elements representing the rebar to provide the necessary contact between the different bars.

3.7 Ground

A *Rigidwall is used to represent the ground along with a rigid shell part for visualization.

4 Additional Files

The following files are required in addition to the main JavaScript

JS Modules Folder

- blast_walls.js
- control_database.js
- element_centroid.js
- end_nodes.js
- facing_segments.js
- geometry.js
- GUI_concrete.js
- GUI_damp_selection.js
- GUI_dr_selection.js
- GUI_general_functions.js
- GUI_material_selection.js
- GUI_spring_selection.js
- GUI_steel.js
- log.txt
- material_read.js
- mesh_channel.js
- mesh_charge.js
- mesh_con_rectangle.js
- mesh_con_round.js
- mesh_hss_rectangle.js
- mesh_hss_round.js
- mesh_isec.js
- vector.js
- viper_segments.js

Damp Properties Folder

- damping_global.csv

Dynamic Relaxation Folder

- dr_prop.csv

Material Properties

- concrete_materials.csv
- rebar_materials.csv
- steel_materials.csv
- example_steel_curve_01.key
- example_steel_curve_01.key

Spring Properties

- bottom_spring.csv
- top_spring.csv

Macros Folder

- area_001.prm
- extrude_001.prm
- hemisphere_001.prm
- line_mesh_001.prm
- merge_001.prm
- merge_all_001.prm
- plate_001.prm
- reflect_part_001.prm
- rotate_node_001.prm
- rotate_part_001.prm
- segment_coat_001.prm
- shell_coat_001.prm
- sphere_001.prm
- translate_node_001.prm
- translate_part_001.prm

Images Folder

- hss_rec.png
- hss_ro.png
- hss_sq.png
- isec.png
- csec.png
- angle.png