

# Gen 2010 (v2.1) Release Note

Integrated Design System for Building and General Structures



## Enhancements

- **Pre/Post Processing** ..... 3
  - (1) Dynamic Report Generation
  - (2) Revit Structure 2010 Interface
  - and much more...
  
- **Analysis** ..... 21
  - (1) Analysis Stop Option in the Pushover Analysis
  - (2) Improved Pushover Analysis Results
  - and much more...
  
- **Design** ..... 26
  - (1) Deflection Check considering Cracked Section
  - (2) Limiting Rebar Ratio
  - (3) Limiting Minimum Section Size
  - and much more...

## □ List of Detailed Enhancements in Pre & Post Processing

- (1) Dynamic Report Generation
- (2) Revit Structure 2010 Interface
- (3) Pressure Type of Beam Loads
- (4) Improved Eccentricity Option in the Element Beam Load and Line Beam Load
- (5) Improved India Standard Section DB
- (6) Changes in the Default Values of Stiffness Scale Factor in the Composite Section for Construction Stage
- (7) Addition of Dimension Import

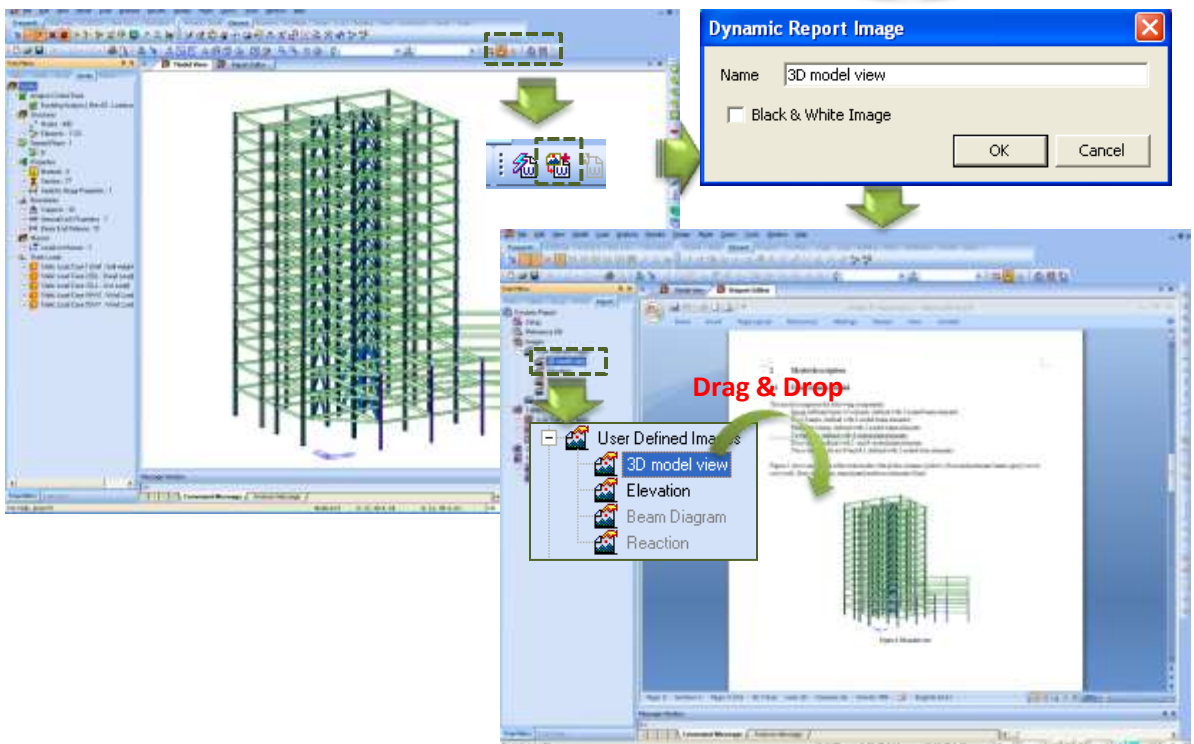
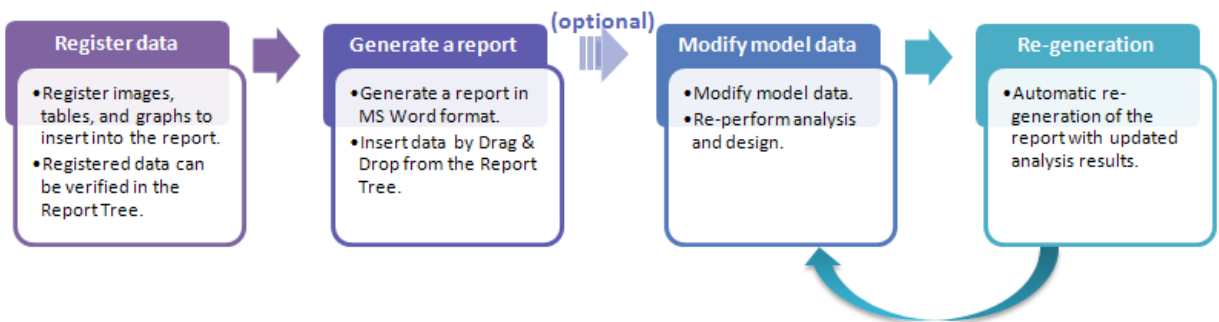
## 1. Dynamic Report Generation

- Word format reports can automatically be generated for selected input & output data (figures, tables, graphs, and text).
- Using “Dynamic Report Regenerator” function, changes of a model file are automatically updated in the report.
- User defined report format can be used and saved.

**Tools > Dynamic Report Generator**

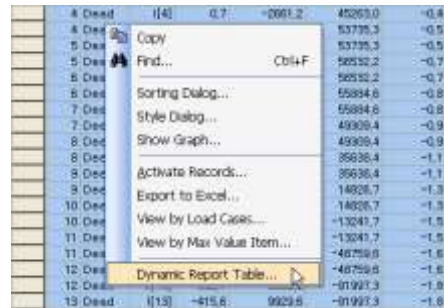
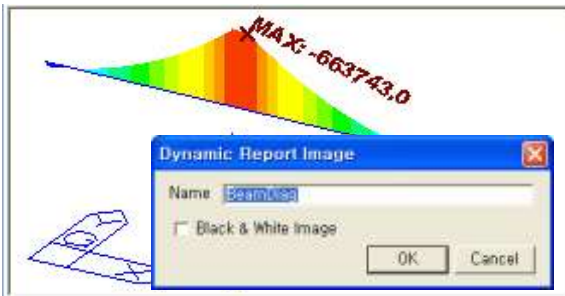
**Tools > Dynamic Report Image**

**Tools > Dynamic Report Auto Generation**



## Procedure for Dynamic Report Generation

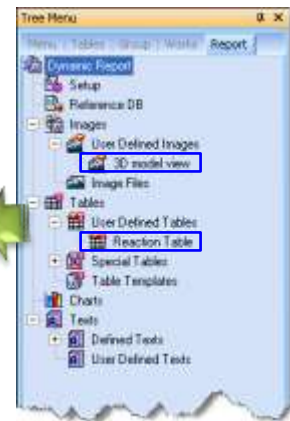
- Step 1. Open a midas Gen model file.
- Step 2. Register contents (images, tables, text summary...) to be entered in the report.  
Registered contents are displayed in the Report Tree.
- Step 3. Open a new report.
- Step 4. Insert the contents by Drag & Drop from the Report Tree.
- Step 5. Modify the report file in the Report Editor and save it in MS word format.



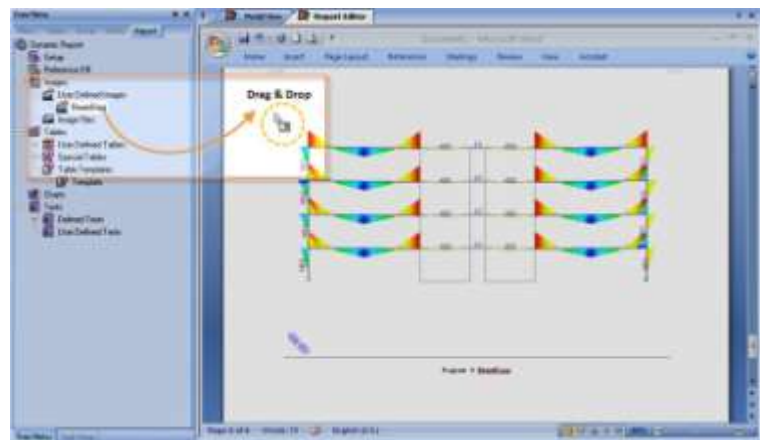
Register the desired data



Open a new report



Report Tree



Insert contents by Drag & Drop

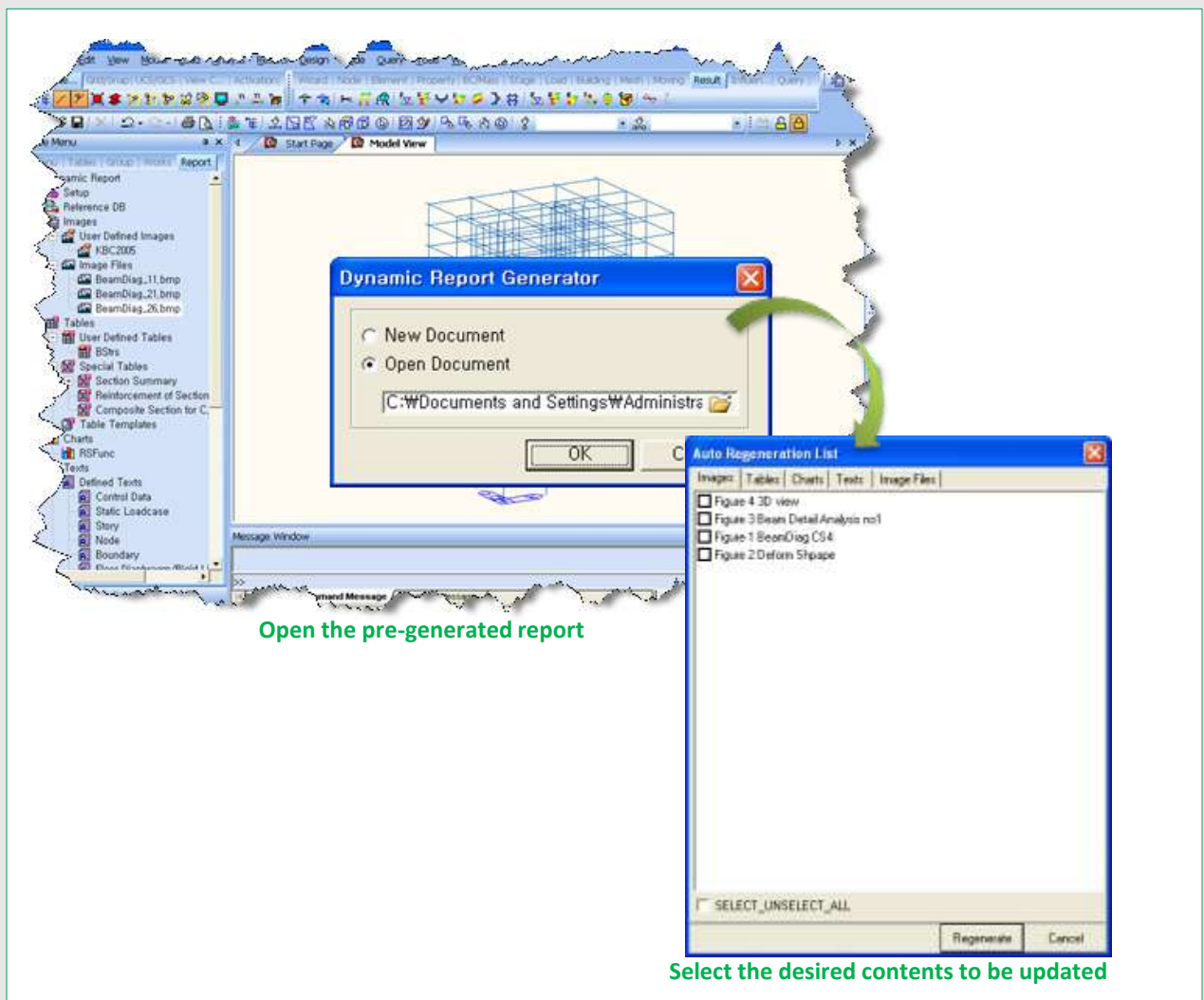
## Procedure for Auto Regeneration

If there are any changes in the model file, we can automatically update the pre-generated report. If the user manually entered additional text or images into the report, those data will remain.

Step 1. Select **Tools > Dynamic Report Image** from the Main Menu, or click icon to open the Auto Regeneration List dialog box.

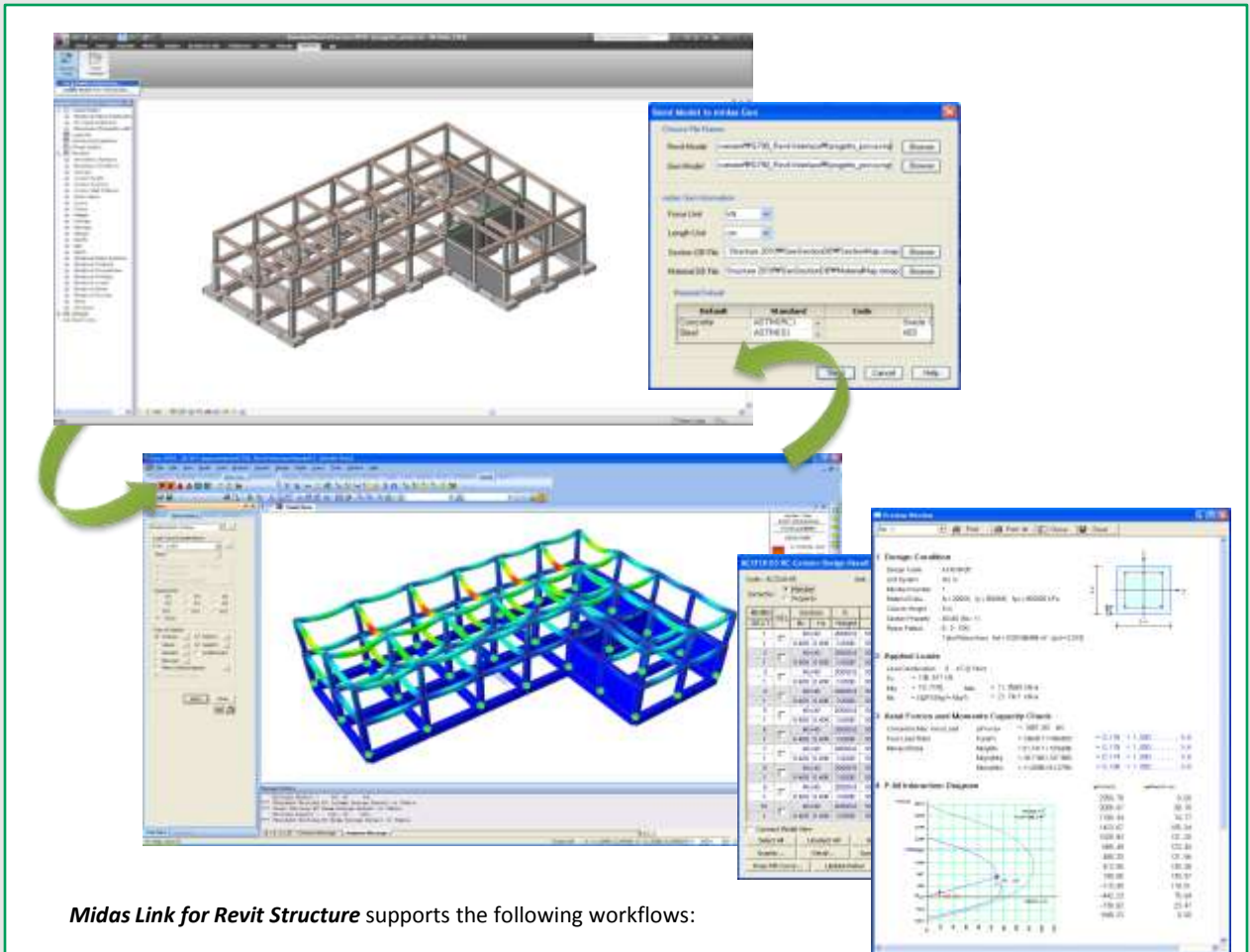
Step 2. All the entered data will be displayed in each tab by data formats. Select the desired data to be updated.

Step 3. Click [**Regenerate**] button.



## 2. Revit Structure 2010 Interface

Using **Midas Link for Revit Structure**, direct data transfer between **midas Gen** and **Revit Structure 2010** is available for Building Information Modeling (BIM) workflow. **Midas Link for Revit Structure** enables us to directly transfer a Revit model data to midas Gen, and delivery back to the Revit model file. It is provided as an **Add-In** module in Revit Structure and midas Gen text file (\*.mgt) is used for the roundtrip.



**Midas Link for Revit Structure** supports the following workflows:

- (1) Send the Revit Structure analytical model to midas Gen.
- (2) Import the MGT file of the Revit model in midas Gen.
- (3) Export the midas model file to the MGT file.
- (4) Update the Revit Structure model from midas Gen

## Getting Started

The following exercise leads you through a sample scenario providing detailed instructions for each step in the process.

### Step 1: Prepare a Model

1. Using the structural template, build a simple structure that has 4 beams, 4 columns, 4 boundary conditions, 1 point load, and 1 line load with host, as shown.
2. Create a load combination; create a load combination usage called MIDAS\_STEEL; set the load combination to this usage.
3. Check for inconsistency by clicking Tools menu > Analytical Model > Analytical / Physical Model Consistency Checks. Make any adjustments if necessary.



### Step 2: Send Model to midas Gen

1. Click Tools menu > External Tools > Send Model to midas Gen
2. Click OK to send the project using the default.

The following table shows the Revit components that are sent to midas Gen. It also shows the corresponding MIDAS commands used in the MGT file.

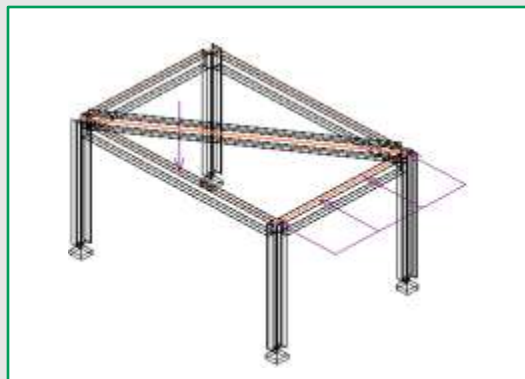
Revit Structure	midas Gen MGT
Level	STORY
Grid	GRIDLINE
Wall	ELEMENT (WALL), NODE
Frame Element (beam, brace)	ELEMENT (BEAM) - multiple if split, NODE, FRAME-RLS
Column	ELEMENT (BEAM) - multiple if split, NODE, FRAME-RLS
Footing	CONSTRAINT – rigid
Boundary Condition (Point)	CONSTRAINT, SPRING
Load Case	STLDCASE
Load Combination (with usage name beginning with MIDAS_)	LOADCOMB
Point Load (on frame element or column ends)	CONLOAD
Point Load (on frame element or column interior)	BEAMLOAD (CONLOAD)
Line Load (on single frame element or column)	LINELOAD, PRESSURE
Area Load (fully enclosed by frame elements or columns)	FLOORLOAD
Rigid Link	RIGIDLINK
Material	MATERIAL
Section	SECTION
Wall Thickness	THICKNESS

### **Step 3: Analyze and Revise Model in midas Gen**

1. Start a new project in midas Gen, and import the MGT file of the Revit model. Note any warnings and errors in the message window.
2. Import the MGT file again, it should import without error and display the structure.
3. Perform a structural analysis in order to be sure that the model is complete.
4. Add a horizontal brace to the model in section 1.
5. Reduce the size of the columns from W10x49 (section 8) to W10x33 (section 9).
6. Export the model using the same name as the imported MGT file.

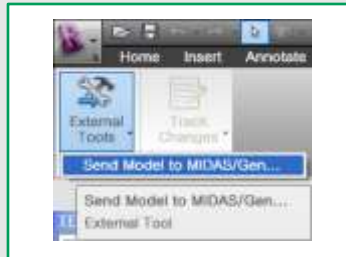
### **Step 4: Update Model from midas Gen**

1. In Revit Structure, click Tools menu > External Tools > Update Model from midas Gen
2. Verify that the paths in the dialog point to the right files, and click Update. The summary should report 1 added element and 4 section changes, and the structure should now look like the following illustration.

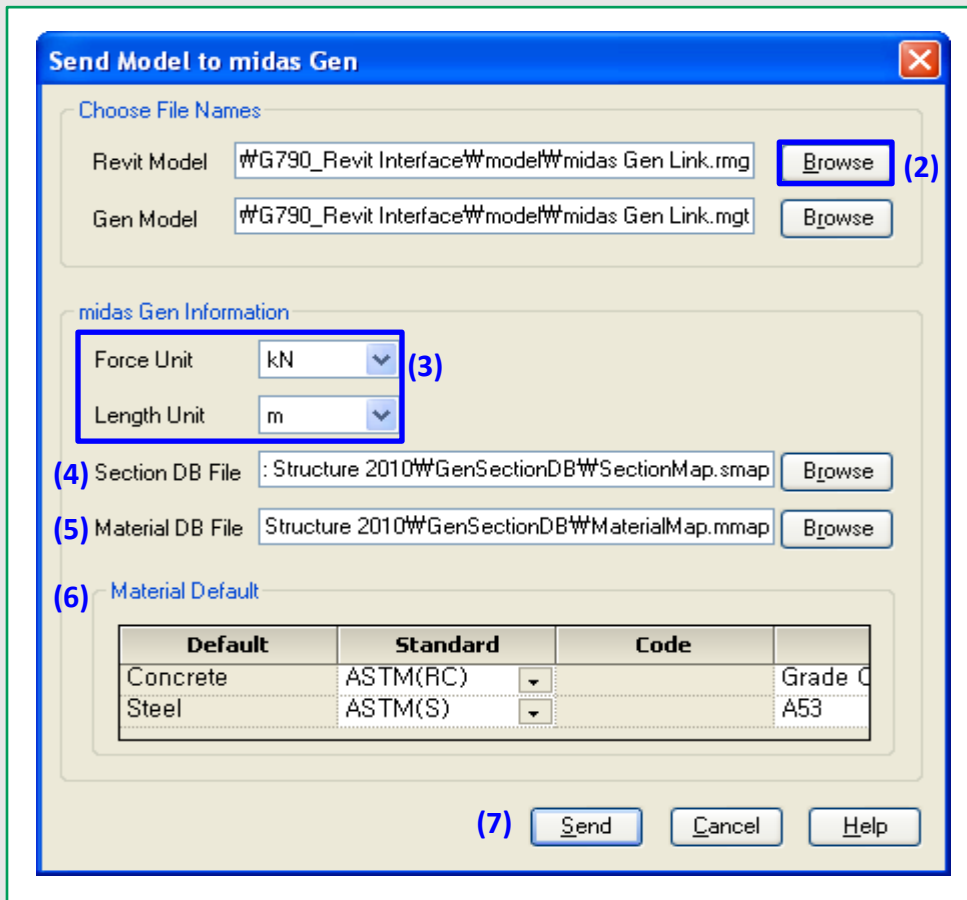


## Send Model to midas Gen

1. In order to send the Revit Structure analytical model to midas Gen, click **Add Ins > External Tools > Send Model to MIDAS/Gen**.



2. Click the first Browse button to select a path for the Revit model file. The paths for the MIDAS model and the log file will default to the same name. If necessary, modify these paths by clicking the corresponding Browse buttons.

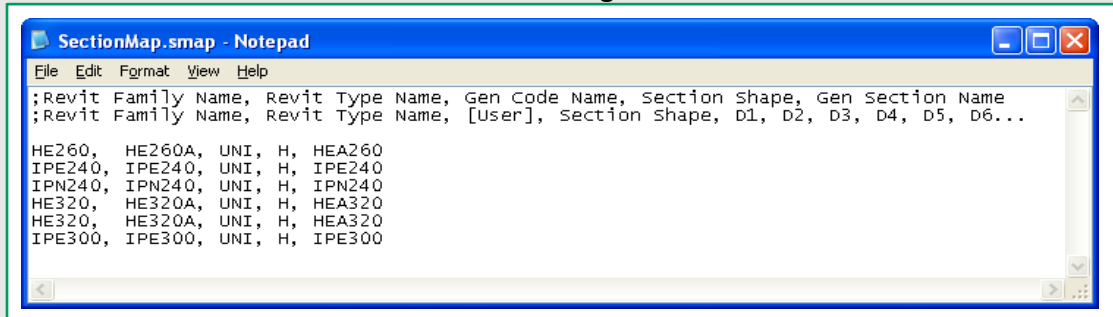


3. Under the **midas Gen Information** group, select the units for length and force to be used in midas Gen.

#### 4. Section DB File

*SectionMap.smap* file is a section transfer list file from Revit Structure to midas Gen. The user can directly add or modify *SectionMap.smap* file to specify section shapes and size imported in midas Gen.

Section transfer data are written in the following formats:



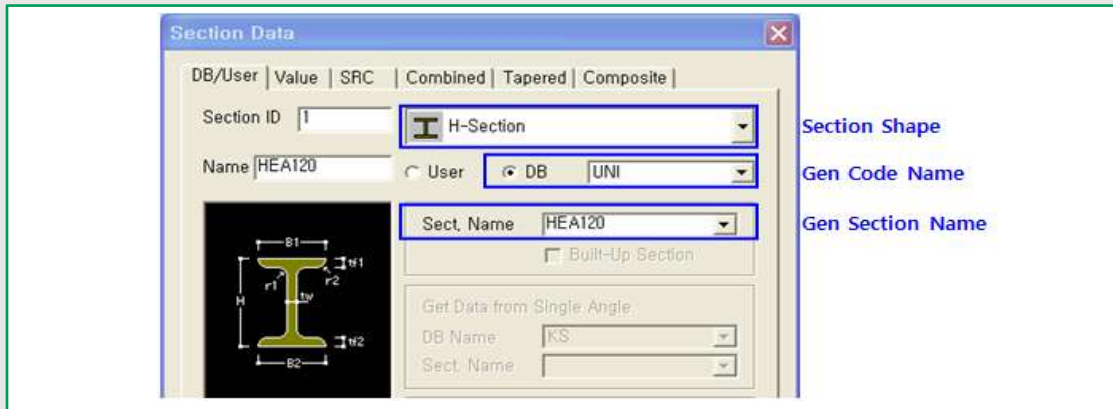
**Revit Family Name:** Family Name used in Revit Structure

**Revit Type Name:** Element type name used in Revit Structure

**Gen Code Name:** Section DB code which will be used in midas Gen

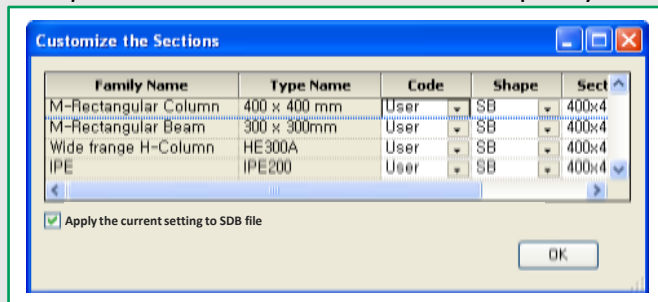
**Section Shape:** Section shape name which will be used in midas Gen (ex. H, L, C, T, P...)

**Gen Section Name:** Section name in the specified section DB code



If an unidentified section exists in a Revit model, the following dialog box is displayed to specify the section shape and size imported in midas Gen. When “Code” is selected in the following dialog box, only “SB (Solid Box)” can be selected for Shape.

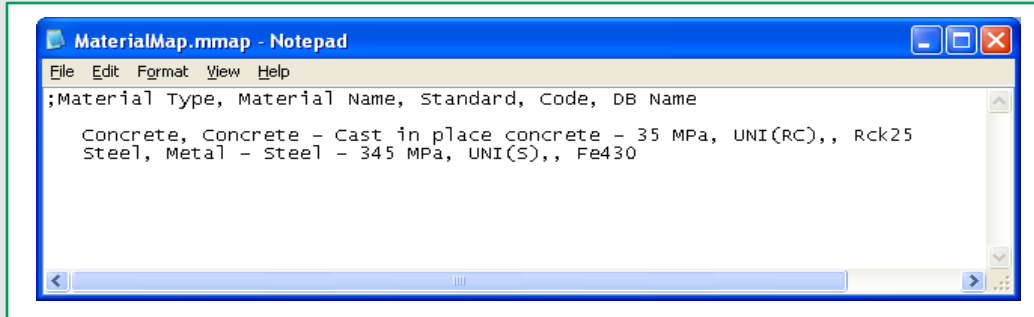
If **Apply the current setting to SDB file** option is checked on, the setting will be saved in *ectionMap.smap* so that the user do not need to specify them again.



## 5. Material DB File

*MaterialMap.mmap* file is a material transfer list file from Revit Structure to midas Gen. The user can directly add or modify *MaterialMap.mmap* to specify material properties imported in midas Gen.

Section transfer data are written in the following formats:



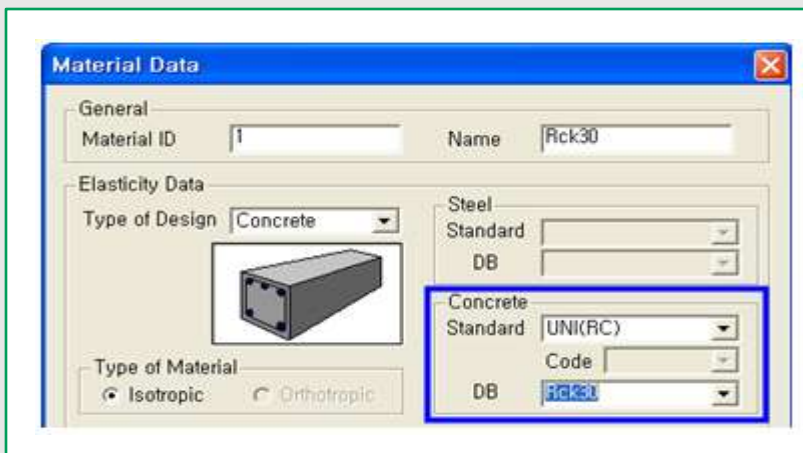
**Material Type:** Material type used in Revit Structure

**Material Name:** Material name used in Revit Structure

**Standard:** Material standard which will be used in midas Gen

**Code:** Concrete design standard for considering the change of modulus of elasticity. This field is required only when the *Standard* is specified as *KS(O1RC)*.

**DB Name:** Material DB name which will be used in midas Gen



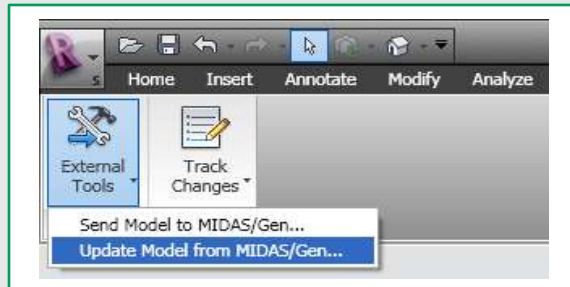
6. Select a default material DB. If the material data is not specified in the Material DB File, default material will be applied in midas Gen.

7. Click **Send** to start writing the MIDAS model.

When the writing is complete, a summary will be presented in a dialog. If the summary indicates issues, further details are available by clicking View Log File.

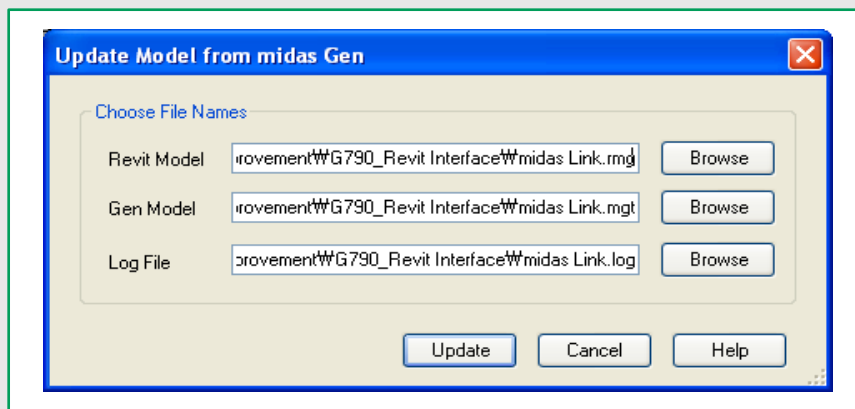
## Update Model from midas Gen

1. In order to update the Revit Structure model from midas Gen, click **Add Ins > External Tools > Update Model from MIDAS/Gen**.



2. A dialog will pop up. If "**Send Model to MIDAS/Gen**" has been used during this session, the paths for the files are automatically set to be the same. If necessary, click Browse to change them.

3. Click Update to start updating the model.



## Applicable data for *MIDAS Link for Revit Structure*

Category	Features	Revit to midas Gen	Remark	
Material	Concrete	v		
	Steel	v		
	Pre Cast Concrete	v		
	Wood	N/A		
	Glass	N/A		
	Ston	N/A		
	Metal	N/A		
Section	Concrete	v		
	Steel	v		
	SRC	N/A		
Member	Column	Vertical Column	v	Only solid rectangular section is applicable.
		Inclined Column	v	
	Beam	Straight Beam	v	
		Curved Beam	N/A	
		Inclined Beam	v	
	Wall	Straight Wall	v	
		Curved Wall	N/A	
		Inclined Wall	N/A	
		Masonry Wall	N/A	
		Wall Opening	N/A	
	Brace	v		
	Truss(Top chord, Bottom chord, and Web)	v		
	Slab	N/A		
Foundation	N/A			
Boundary	Support(Hinge, Roller, Fixed)	v		
	Beam End Release	v		
	Section Offset	N/A		
Static Load	Self Weight	N/A	Load Nature Name : Dead	
	Dead Load	v	Load Nature Name : Dead	
	Live Load	v	Load Nature Name : Live	
	Wind Load	v	Load Nature Name : Wind	
	Seismic Load	v	Load Nature Name : Seismic	
	Temperature Load	v	Load Nature Name : Temperature	
	Snow Load	v	Load Nature Name : Snow	
	Accidental Load	v	Load Nature Name : Accidental	
	Live Load on the roof	v	Load Nature Name : Roof Live	
	Point Load , Hosted Point Load <sup>1)</sup>	v		
	Line Load , Hosted Line Load <sup>1)</sup>	v		
	Area Load <sup>1), 2)</sup>	v		
Hosted Area Load <sup>1)</sup>	N/A			
Load Combination	Load Combination <sup>3)</sup>	v		

### Note

- 1) In order to export point loads, line loads, or area loads to midas Gen, those loads need to be separately entered for each element. For example, if we enter a line load to continuous beam (element no. 1 and 2), we need to enter the load to element no. 1 and 2 separately.
- 2) Area Load in Revit Structure is imported as Floor Load (Polygon-Length type) in midas Gen.
- 3) The name of *Load Combination Usage* specified in Revit Structure will determine the load combination tab in midas Gen. When the name of Load Combination Usage is entered as Steel, Concrete, SRC or Footing, the combination shall be included in the Steel Design, Concrete Design, SRC Design or Footing Design tab of Load Combination dialog box in midas Gen. For the other names of *Load Combination Usage*, the corresponding load combinations will be included in General tab of Load Combination dialog box.

## **What is Updated from midas Gen to Revit Structure**

Here are the revisions that are detected and updated in the Revit model:

### **Sections**

- If assigned section is changed to a pre-defined section in the model, the corresponding element in Revit will be updated accordingly.
- If assigned section is changed to a new section in midas Gen, the corresponding element in Revit will be assigned to a default section (arbitrary section which has a same material type in a model).

### **Delete Elements**

If an element is deleted in midas Gen, the corresponding element in Revit will be deleted accordingly.

### **Move Elements**

If an element is moved in midas Gen, the corresponding frame element or column in Revit will be moved accordingly.

### **Add Elements**

If a beam element (solid box section only) is newly added, a corresponding element in Revit will be added accordingly.

### **Change Beta-Angle**

If beta-angle in a beam element is changed, a corresponding element in Revit will be updated accordingly.

### **Materials**

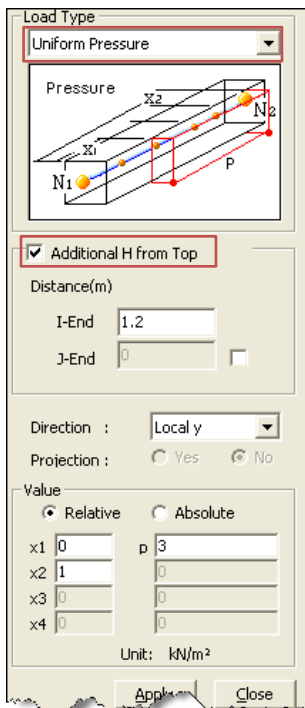
If material data assigned to an element is modified, a corresponding element in Revit will be assigned to a default material (arbitrary material existed in Revit).

### 3. Pressure Type of Beam Loads

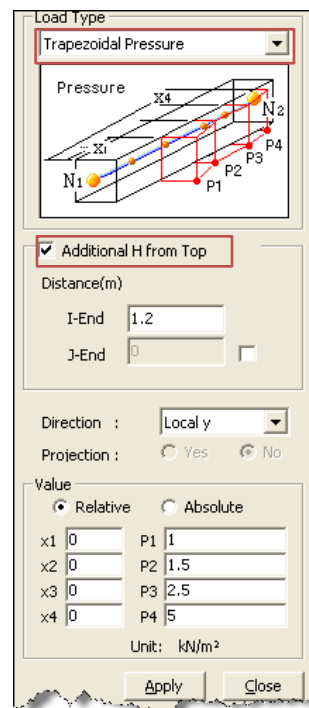
- “Uniform Pressure” and “Trapezoidal Pressure” type of Line Beam Loads and Element Beam Loads have been implemented to consider the width of beam elements when entering wind loads.
- This feature is useful to assign wind loads to the tapered girders (ex. longitudinal girders in bridges).

#### Loads > Element Beam Loads

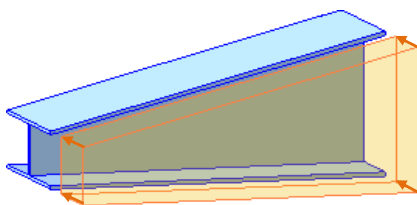
#### Loads > Line Beam Loads



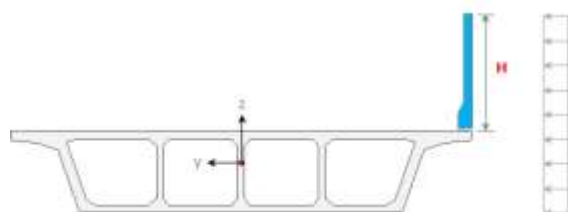
Uniform Pressure type



Trapezoidal Pressure type



Assign a beam load as a pressure load considering the beam width



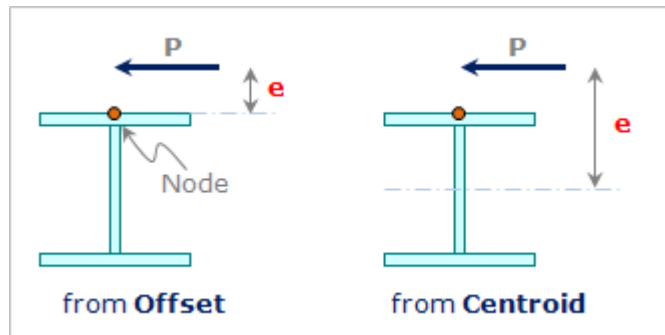
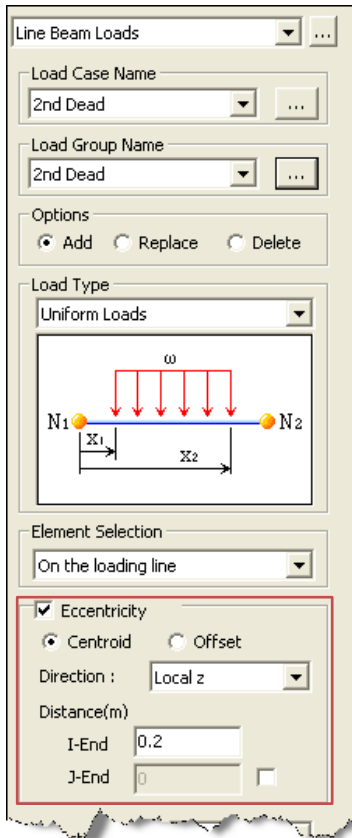
Additional H to consider the additional height of the structures which was not included in the modeling (ex. guard fence)

### 4. Improved Eccentricity Option in the Element Beam Load and Line Beam Load

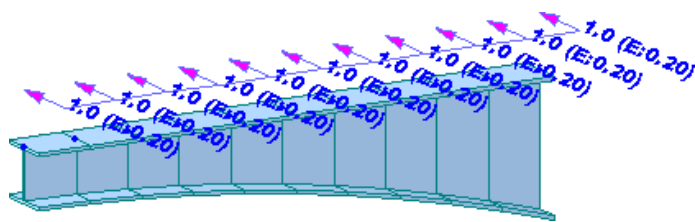
- Eccentricity can be entered by the following 2 methods: 1) from the centroid, and 2) from the offset point.
- In the previous version, the eccentricity was entered only from the centroid. In case of the tapered section, beam loads with eccentricity can be easily entered from the offset point.

Loads > **Element Beam Loads**

Loads > **Line Beam Loads**



Two methods to enter the eccentricity

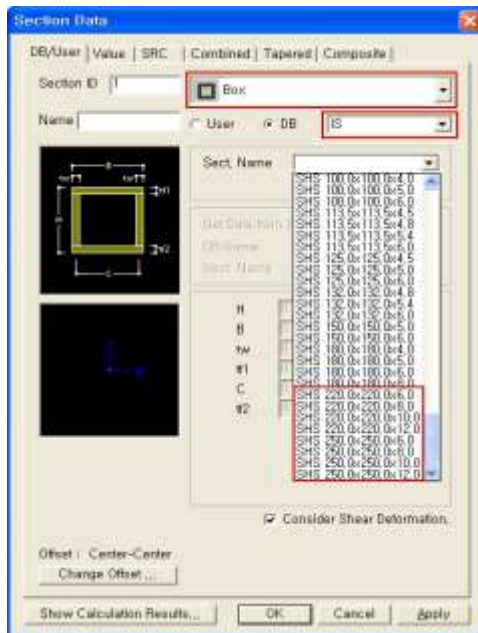


Enter the eccentricity *from Offset* (nodal position) for the tapered sections

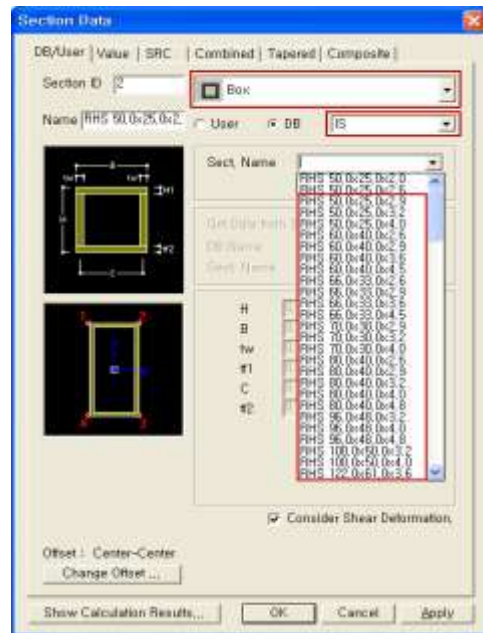
## 5. Improved India Standard section DB

- New sections (Box, Pipe, and Angle) have been implemented in IS, IS1161, and IS808 section DB.

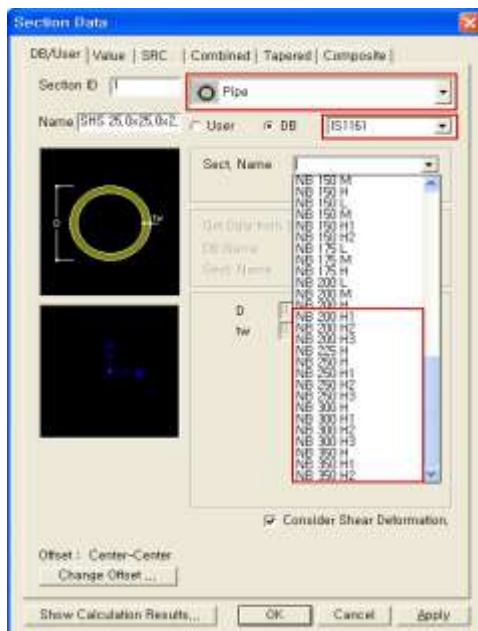
**Model > Properties > Section**



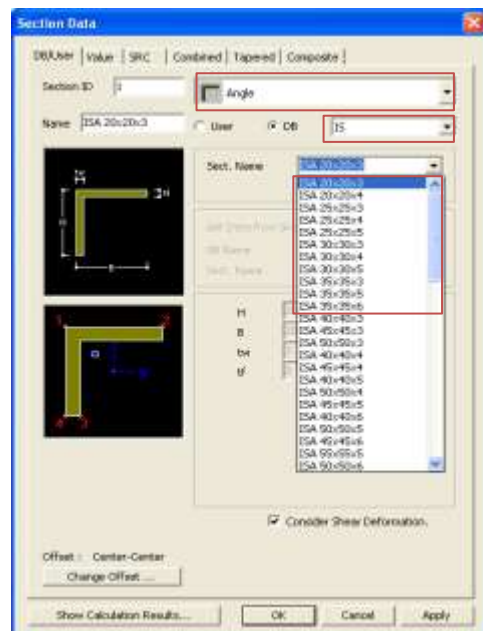
**Box Section SHS**



**Box Section RHS**



**Pipe Section NB**



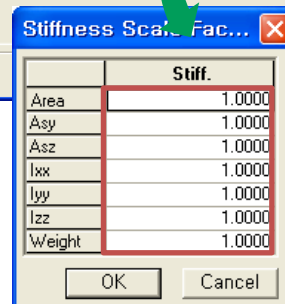
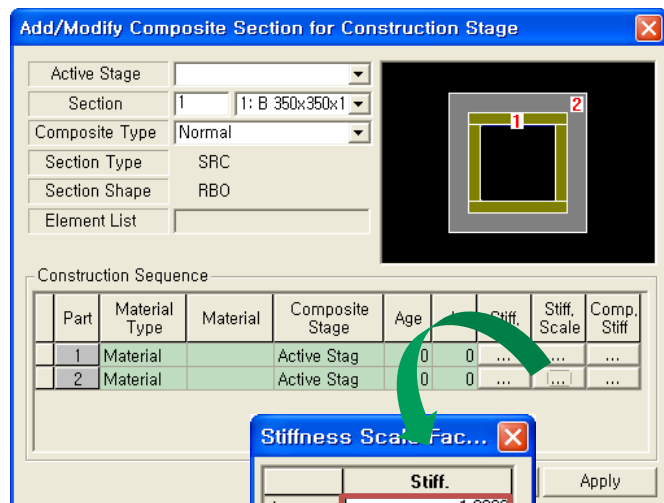
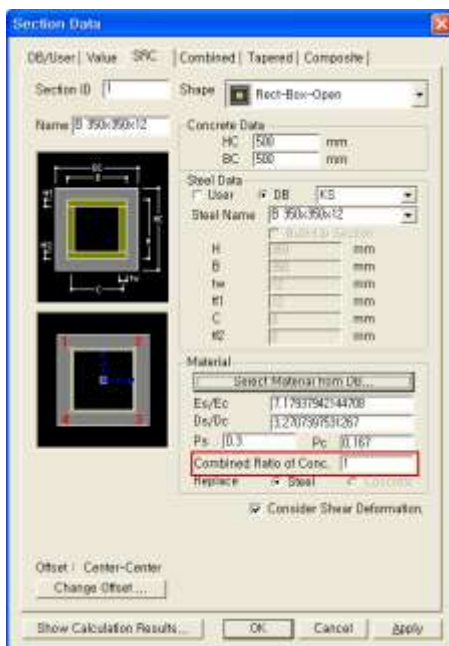
**Angle Section Data**

## 6. Changes in the Default Values of Stiffness Scale Factor in the Composite Section for Construction Stage

- In the SRC type section dialog box, the default value of “Combined Ratio of Conc.” has been changed from “0.8” to “1.0” in order to prevent user’s confusion.
- In the Composite Section for Construction Stage dialog box, the default values of “Stiffness Scale Factor” have been changed from “0.8” to “1.0”.

**Model > Properties > Section**

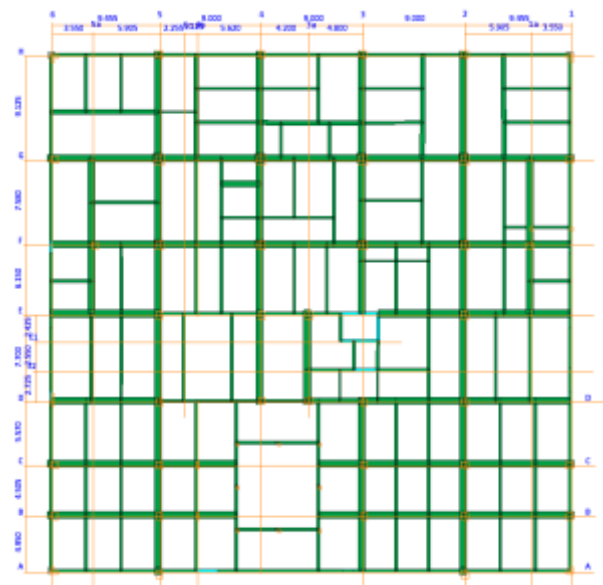
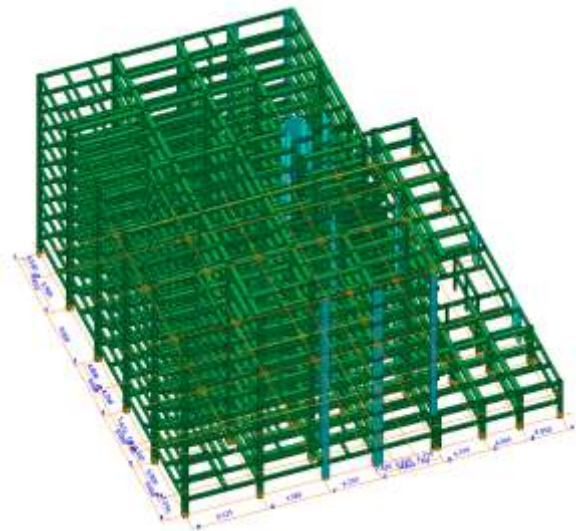
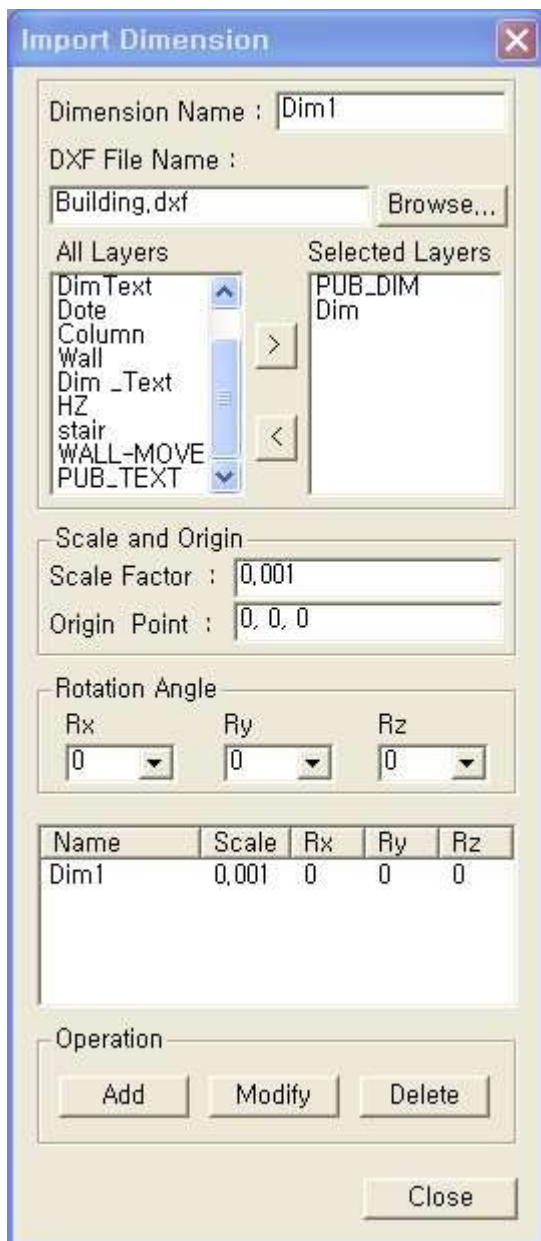
**Load > Construction Stage Analysis Data > Composite Section for Construction Stage**



## 7. Addition of Dimension Import

- Import dimension layers from AutoCAD DXF. Dimensions are displayed as guide lines on the screen such as grid lines and it does not affect analysis results.

### Model > Dimension



## □ List of Detailed Enhancements in Analysis

- (1) Analysis Stop Option in the Pushover Analysis
- (2) Improved Pushover Analysis Results
- (3) Mander Model in the Inelastic Material Properties
- (4) Improved MINEGOTTO-PINTO Steel Model

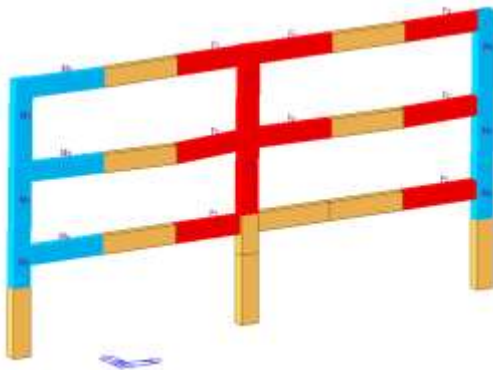
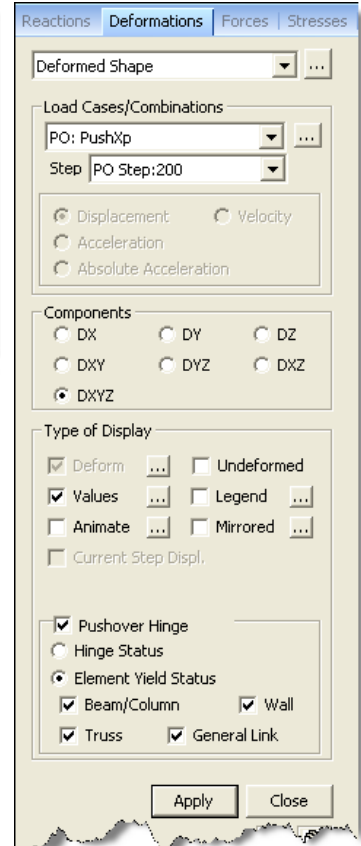
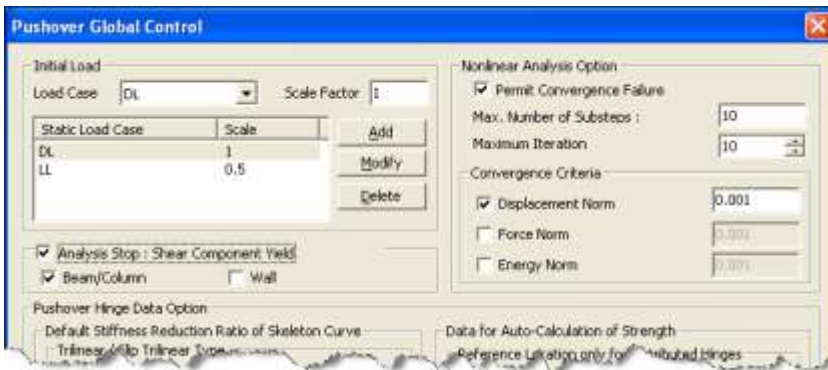
## 1. Analysis Stop Option in the Pushover Analysis

- **Analysis Stop option in the Pushover Global Control:** When a shear hinge occurs in the selected member, the pushover analysis will be automatically terminated. In this case, the analysis results can be examined up to the last pushover step.
- **Element Yield Status display in the Deformed Shape:** The yield status of components (Fx, Fy & Fz, Mx, and My & Mz) by pushover analysis is produced. This option is useful in verifying if a shear failure occurred in any element.
- **Shear Yield Element table:** The elements for which shear hinge occurred prior to the moment hinges or axial hinges are plotted in a spreadsheet format table.

*Design > Pushover Analysis > Pushover Global Control*

*Results > Deformations > Deformed Shape*

*Design > Pushover Analysis > Pushover Hinge Result Table > Shear Yield Element*



midas Gen  
POST-PROCESSOR  
**DEFORMED SHAPE**  
**ELEMENT YIELD STATUS**

	5.6%	Fy, Fz
	0.0%	Fx
	5.0%	My, Mz

SCALE FACTOR=  
1.0000E+000

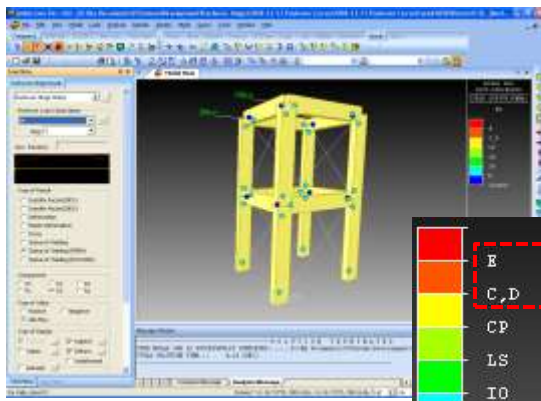
## 2. Improved Pushover Analysis Results

- Pushover analysis results have been graphically improved in order to prevent the user’s confusion.
- In FEMA and Eurocode type hinge, the hinge status of “C” point and “D” point is now displayed separately. In the previous version, the hinge status of “C,D” was displayed after the C point.

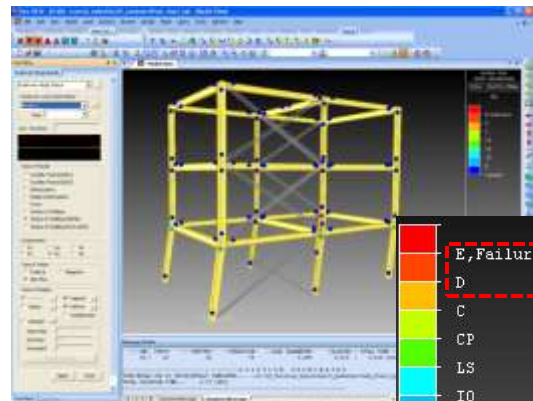
Result > Deformations > Deformed Shape

Design > Pushover Analysis > Pushover Hinge Status Result

Design > Pushover Analysis > Pushover Hinge Result Table > Beam Summary



Previous version



New version

### Pushover Hinge Status Result

Step	Deform	Force	Plastic Deform	Initial Str	D/D1	D/D2	Status
139	1.195e-003	34665.670	0.015e-004	88057971.014	3.036e+000		CP
140	1.263e-003	34665.670	0.981e-004	88057971.014	3.264e+000		CP
141	5.259e-003	6933.134	4.885e-003	88057971.014	1.336e+001		Failure
142	5.301e-003	6933.134	4.907e-003	88057971.014	1.346e+001		Failure
143	5.341e-003	6933.134	4.947e-003	88057971.014	1.357e+001		Failure
144	5.390e-003	6933.134	4.986e-003	88057971.014	1.367e+001		Failure
145	5.418e-003	6933.134	5.024e-003	88057971.014	1.378e+001		Failure
146	5.515e-003	6933.134	5.121e-003	88057971.014	1.401e+001		Failure
147	5.582e-003	6933.134	5.188e-003	88057971.014	1.418e+001		Failure
148	1.197e-002	0.000	1.158e-002	88057971.014	3.042e+001		Failure
149	1.198e-002	0.000	1.158e-002	88057971.014	3.042e+001		Failure

Previous version

Plastic Deform	Initial Str	D/D1	D/D2	Status
0.015e-004	88057971.014	3.036e+000		CP(+)
0.981e-004	88057971.014	3.264e+000		CP(+)
4.885e-003	88057971.014	1.336e+001		D(+)
4.907e-003	88057971.014	1.346e+001		D(+)
4.947e-003	88057971.014	1.357e+001		D(+)
4.986e-003	88057971.014	1.367e+001		D(+)
5.024e-003	88057971.014	1.378e+001		D(+)
5.121e-003	88057971.014	1.401e+001		D(+)
5.188e-003	88057971.014	1.418e+001		D(+)
1.158e-002	88057971.014	3.042e+001		E, Failure
1.158e-002	88057971.014	3.042e+001		E, Failure

New version

### Beam Summary Table

### 3. Mander Model in the Inelastic Material Properties

- Mander Model has been added to model the behavior of concrete confined with steel stirrups.
- The menu name, **Fiber Material Properties**, has been changed to **Inelastic Material Properties**.

**Model > Properties > Inelastic Material Properties**

The screenshot shows the 'Inelastic Material Model' dialog box with the following settings:

- Name: Concrete
- Material Type: Concrete
- Hysteresis Model: Mander Model
- Concrete Type:  Confined Concrete : Core Concrete
- Section & Confinement Rebar Type:  Rectangular
- Section Data: Acc: Area of Effective Concrete Core (1800000.0000 mm²), Ae: Total Area of Effectively Confined Core Concrete (820000.0000 mm²), ke = Ae / (Ac (1-ρcc)) = Ae / Acc = 0.455556 < 1.0
- Unconfined Concrete Data: f'c: Unconfined Concrete Strength (40.0000 N/mm²), ε'c: Unconfined Concrete Strain (0.002), Ec: Elastic Modulus of Concrete (Mander et al., 31622.7766 N/mm²), f't: Tensile Strength of Concrete

The 'Mander Material Data' graph shows Compressive Stress,  $f_c$ , versus Compressive Strain,  $\epsilon_c$ . It compares the behavior of Unconfined Concrete and Confined Concrete. Key points on the graph include  $f_{c0}$ ,  $f_{cc}$ ,  $\epsilon_{c0}$ ,  $2\epsilon_{c0}$ ,  $\epsilon_{cp}$ , and  $\epsilon_{cr}$ . The graph also indicates the elastic modulus  $E_c$  and the tensile strength  $f_t$  of the concrete.

Below the graph, the following formulas are provided:

- Elastic modulus of Concrete :  $E_c = 5,000 \sqrt{f_{c0}}$  MPa
- Tensile Strength of Concrete :  $f_t = 0.62 \sqrt{f_{c0}}$  MPa
- Tensile Strain of Concrete :  $\epsilon_t = \frac{f_t}{E_c}$

The 'Material...' menu is open, showing various material models. The 'Inelastic Material Properties...' option is highlighted with a red box.

### 4. Improved MINEGOTTO-PINTO Steel Model

- In MINEGOTTO-PINTO steel model, the equation for calculating the plastic strain,  $\xi$ , has been changed in order to enhance the convergence performance.

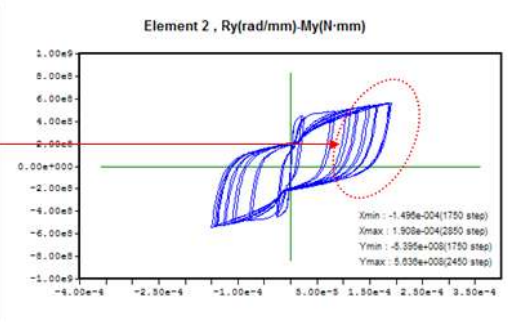
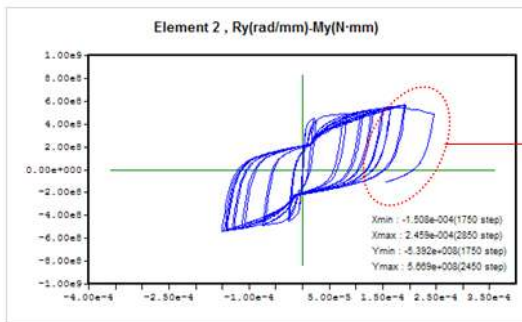
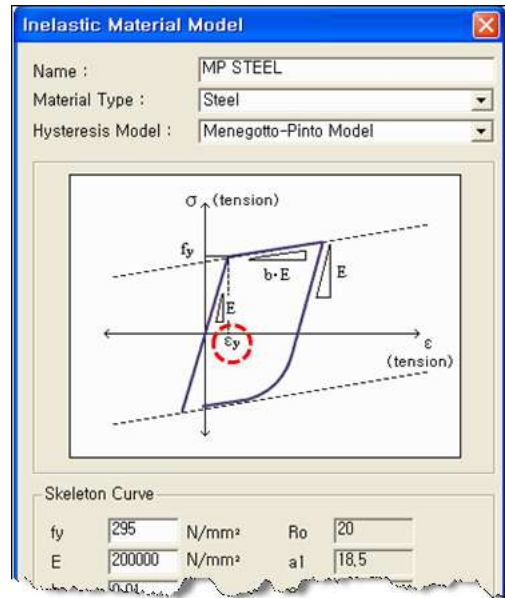
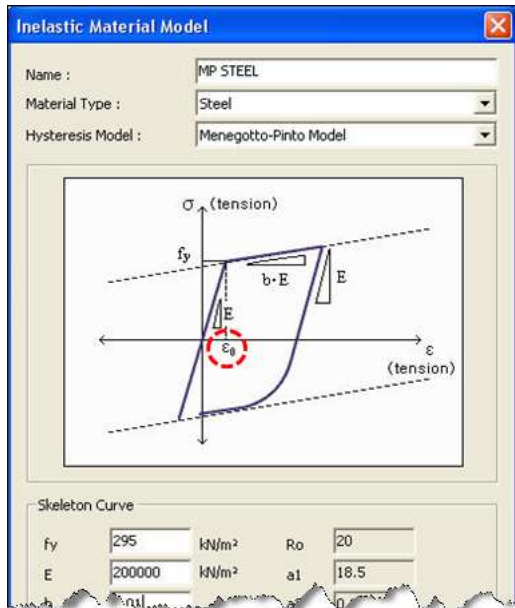
**Model > Properties > Inelastic Material Properties**

**Previous version**

$$\xi = \begin{cases} \Delta\varepsilon > 0 ; \xi_1 = \frac{\varepsilon_{\max} - \varepsilon_0}{\varepsilon_0 - \varepsilon_r} \\ \Delta\varepsilon < 0 ; \xi_2 = \frac{\varepsilon_0 - \varepsilon_{\min}}{\varepsilon_0 - \varepsilon_r} \end{cases}$$

**New version**

$$\xi = \begin{cases} \Delta\varepsilon > 0 ; \xi_1 = \frac{\varepsilon_{\max} - \varepsilon_0}{\varepsilon_y} \\ \Delta\varepsilon < 0 ; \xi_2 = \frac{\varepsilon_0 - \varepsilon_{\min}}{\varepsilon_y} \end{cases}$$



-----TIME STEPS-----SUBSTEPS-- --ITERATIONS--  
 2950 / 2950 2961 7578

-----TIME STEPS-----SUBSTEPS-- --ITERATIONS--  
 2950 / 2950 2958 7496



## □ List of Detailed Enhancements in Design part

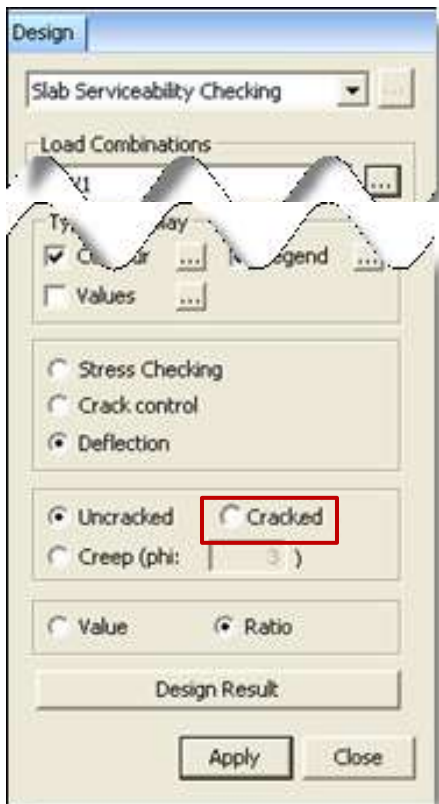
- (1) Slab Deflection Check considering Cracked Section
- (2) Limiting Rebar Ratio
- (3) Limiting Minimum Section Size
- (4) Improved Concrete Code Design as per the Latest Italy NA of Eurocode2:04
- (5) Improved Capacity Design for Walls
- (6) Serviceability Checking as per TWN-LSD96 and TWN-ASD96

## 1. Slab Deflection Check considering Cracked Section

Deflections considering cracked section can be calculated in *Slab Serviceability Checking*. midas Gen performs a cracked section analysis for the generated *Crack Analysis Load Cases*. Deflection check considering long term effect can also be calculated by applying the creep coefficient.

Members which are expected to crack, but may not be fully cracked, will behave in a manner intermediate between the uncracked and fully cracked conditions and an adequate prediction of behavior is given by the equation below based on the sub clause 7.4.3 (3) in EN1992-1-1:2004. Following factors including the effective moment of inertia by elements for each iteration step can be checked in "File Name\_CSA.OUT" file.

**Design > Meshed Slab/Wall Design > Slab Serviceability Checking**  
**Design > Meshed Slab/Wall Design > Perform Cracked Section Analysis**  
**Design > Meshed Slab/Wall Design > Cracked Section Analysis Control**



$$\alpha = \zeta \alpha_{II} + (1 - \zeta) \alpha_I$$

Therefore  $I_{eff}$  (effective moment of inertia) can be calculated from the following equation:

$$\frac{1}{I_{eff}} = \zeta \frac{1}{I_{cr}} + (1 - \zeta) \frac{1}{I_g}$$

Where,

$\zeta$  is a distribution coefficient, given by the following equation:

$$\zeta = 1 - \beta \left( \frac{M_{cr}}{M} \right)^2$$

' $\beta = 0.5$ ' is applied (long – term loading).

$$M_{cr} = \frac{f_{ctm} b h^2}{6}$$

$$I_{cr} = A_s (d - d_c)^2 \frac{E_s}{E_c} + \frac{1}{3} b d_c^3$$

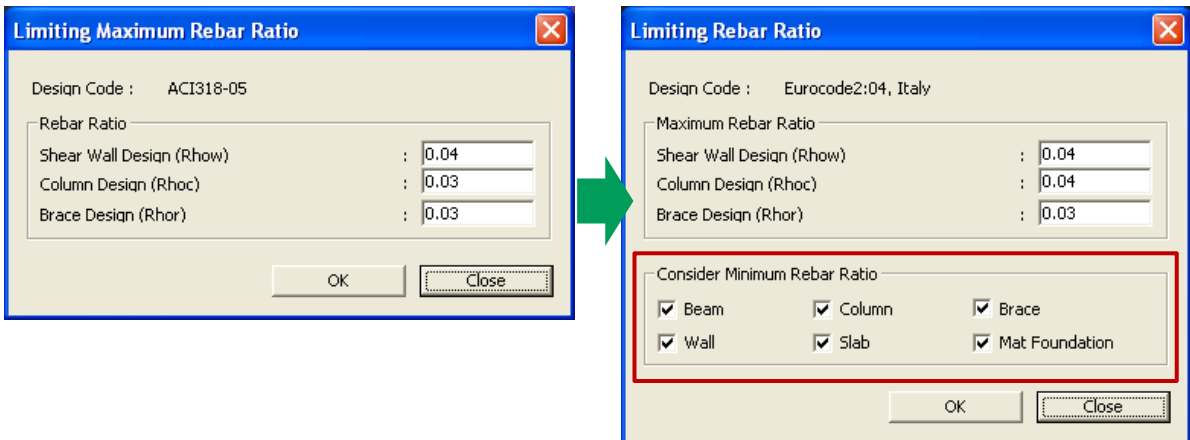
$$d_c = \frac{-A_s E_s + \sqrt{(A_s E_s)^2 + 2 b A_s E_s E_{c,eff} d}}{b E_{c,eff}}$$

## 2. Limiting Rebar Ratio

In concrete code design as per Eurocode2:04 (Italy National Annex) and capacity design as per Eurocode8:04 (Italy National Annex) and NTC2008, the user can select whether to consider minimum rebar ratio limitation.

Limiting Rebar Ratio will be useful to find required rebar ratio regardless the minimum rebar ratio limitation.

### Design > Concrete Code Design > Limiting Rebar Ratio



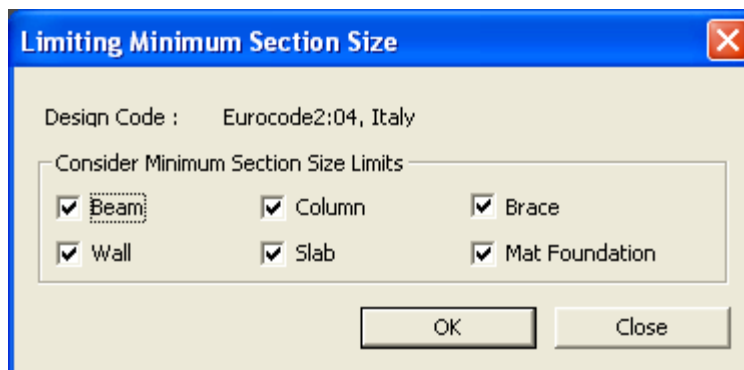
### Applied minimum rebar requirements

Concrete Design		Capacity Design		
	EC2-1-1:04, Italy NA		EC8-1:04, Italy NA	NTC2008
Beam	$p_{min} = \max(0.26 \cdot f_{ctm} / f_y, 0.0013)$ EC2-1-1:04 9.2.1.1(1)	Beam	$p_{min} = 0.5(f_{ctm} / f_{yk})$ EC8-1 5.4.3.1.2 (5)	min : 1.4/fy
Columns	$p_{min} = \max(0.1 N_{ed} / f_{yd}; 0.003A_c)$ EC2-1-1:04 9.5.2(2)	Columns	$p_{min} = 0.01$	min : 0.01
Wall element	$p_{min} = 0.004$	Wall element	1. critical region - $\epsilon_{c} > 0.002$ : 0.005 - $\epsilon_{c} < 0.002$ : 0.002 2. non-critical : 0.002 EC8-1 5.4.3.4.2(11)	1. Within the confined zones: 1% to 4% 2. Outside the confined zones: 0.2%
Slab	$p_{min} = \max(0.26 \cdot f_{ctm} / f_y, 0.0013)$ EC2-1-1:04 9.2.1.1(1)	Secondary seismic elements, beam	$\max(0.26 \cdot f_{ctm} / f_y, 0.0013)$	Same as EC2 Design
Mat Foundation	$p_{min} = \max(0.26 \cdot f_{ctm} / f_y, 0.0013)$ EC2-1-1:04 9.2.1.1(1)	Secondary seismic elements, column	$\max(0.1 N_{ed} / f_{yd}; 0.003A_c)$	Same as EC2 Design

### 3. Limiting Minimum Section Size

In concrete code design as per Eurocode2:04 (Italy National Annex) and IS 13920:1993, and capacity design as per Eurocode8:04 (Italy National Annex) and NTC2008, the user can select whether to consider the minimum section size.

*Design > Concrete Code Design > Limiting Minimum Section Size*



#### □ Applied minimum rebar requirements

##### (1) Eurocode2-1-1:04

- Beam: 120mm
- Column: 120mm
- Wall: 100mm
- Slab ; 200mm for SLS design crack control
- Mat ; 200mm

##### (2) Eurocode8-1:04, NTC 2008

- Beam:  $bw \leq \min\{bc+hw, 2bc\}$  for DCM and DCH  
200mm for DCH
- Column: 250mm
- Wall:  $\max(150\text{mm}, 1/20 \times \text{height})$

##### (3) IS456:2000, IS13920:1993

- Beam:  $w/d \geq 0.3$ ,  $\min\{bc, hc\} \geq 200\text{mm}$ ,  $d \leq 1/4 \times lcr$
- Column:  $\min\{bc, hc\} \geq 200\text{mm}$ ,  $dc/db \geq 0.4$   
If the column height is larger than 5m or unbraced length is larger than 4m, section dimension should be larger than 300mm.
- Wall:  $t \geq 150\text{mm}$

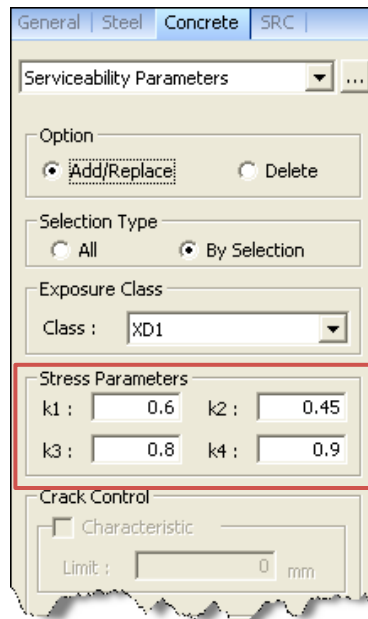
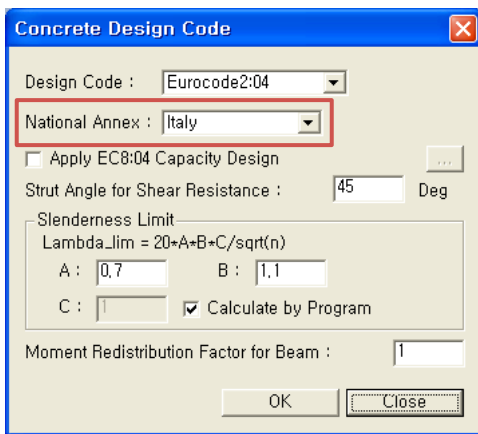
## 4. Improved Concrete Code Design as per the Latest Italy NA of Eurocode2:04

- The default values for the stress parameter (k4) has been changed based on the latest Italy national annex. (1.0 in the previous version, 0.9 in the new version)
- The equation of the strength reduction factor, v, to calculate the shear force has been changed based on the latest Italy national annex.

**Design > Concrete Design Parameter > Design Code**

**Design > Concrete Design Parameter > Serviceability Parameters**

**Design > Concrete Code Design > Beam Design**



### Eurocode2:04, 6.2.2(6)

... The shear force  $V_{Ed}$ , calculated without reduction by  $\beta$ , should however always satisfy the condition

$$V_{Ed} \leq 0,5 b_w d v f_{cd}$$

where v is a strength reduction factor for concrete cracked in shear...

#### Previous version

$$v = 0,7 \left[ 1 - \frac{f_{ck}}{250} \right] \quad (f_{ck} \text{ in MPa})$$

#### New version

$$v = 0,5 \quad \text{fino a C70/85}$$

$$v = 0,7 \left[ 1 - \frac{f_{ck}}{250} \right] \quad \text{per C80/95 e C90/105}$$

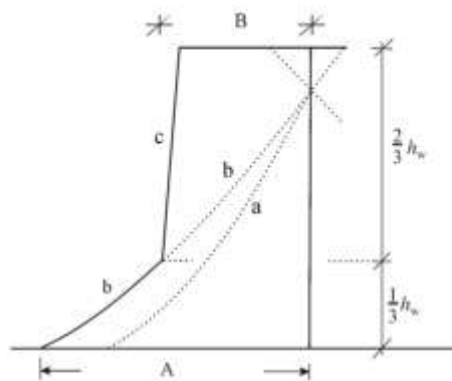
## 5. Improved Capacity Design for Walls

- In the Capacity Design of walls, the magnification factor,  $\varepsilon$ , has been corrected to consider the design bending moment ( $M_{Ed}$ ) and the design flexural resistance ( $M_{Rd}$ ) at the base of the wall. In the previous version the design bending moment ( $M_{Ed}$ ) and the design flexural resistance ( $M_{Rd}$ ) were calculated at the bottom of the walls at each story.

*Design > Concrete Design Parameter > Design Code*

*Design > Concrete Code Design > Wall Design*

**Eurocode8:04, 5.4.2.4, Figure 5.4**



### Key

- a shear diagram from analysis
- b magnified shear diagram
- c design envelope
- A  $V_{wall,base}$
- B  $V_{wall,top} \geq V_{wall,base}/2$

Figure 5.4: Design envelope of the shear forces in the walls of a dual system.

**Eurocode8:04, 5.5.2.4.1, equation (5.25)**

$\varepsilon$  is the magnification factor, calculated from expression (5.25), but not less than 1,5:

$$\varepsilon = q \cdot \sqrt{\left( \frac{\gamma_{Rd}}{q} \cdot \frac{M_{Rd}}{M_{Ed}} \right)^2 + 0,1 \left( \frac{S_e(T_c)}{S_e(T_1)} \right)^2} \leq q$$

## 6. Serviceability Checking as per TWN-LSD96 and TWN-ASD96

- In the Steel Code Check as per TWN-LSD96 and TWN-ASD96, serviceability check has been added for beam and column members.
- In the Serviceability Parameters, the user can specify the limit values for the deflection check.

**Design > Steel Design Parameter > Design Code**

**Design > Steel Design Parameter > Serviceability Parameters**

**Design > Steel Code Check**

Serviceability Parameters

Option  
 Add/Replace  Delete

Selection Type  
 All  By Selection

Deflection Control For Beams  
 L / 360  
 User : L /

Deflection Control For Columns  
 h / 300  
 User : h /

Deflection Amplification Factor

Apply Close

TWN-ASD96 Code Checking Result Dialog

CH	MEMB	SECT	BEL	Section		LCB	Primary Sorting Option												Def
				Material	Fy		Pa	My	Mz	Ob	Iz	Cmp	Fu	Fby	Fbz	Defa			
OK	336	528		S05, W04x100		10.0000	10.0000	10.0000	1.000	1.000	0.0000	187170	0.0000				-0.0124		
OK	1521	531		A36, W12x56	248211	0.0000	0.0000	0.0000	1.000	1.000	1.000	124106	163028	149207			0.0300		
OK	1675	532		A36, W12x56	248211	0.0000	0.0000	0.0000	1.000	1.000	1.000	124106	163028	149207			0.0167		
OK	1498	528		A36, W12x56	248211	0.0000	0.0000	0.0000	1.000	1.000	1.000	124106	163028	149207			-0.1340		
OK	95	531		C0A, W14x130		0.0000	0.0000	0.0000	1.000	0.000	0.000	104001	17382	742616			0.00747		
OK	1069	538		A36, W14x130	413986	-4105.2	58.4275	81.3641	1.000	1.000	0.000	17382	248211	203264			0.01667		
OK	336	552		C0A, W14x88		0.0000	0.0000	0.0000	1.000	0.000	0.000	10672	25283	88158			0.0000		
OK	0.962	0.962		A36, W14x88	413986	-566.54	79.8592	21.4338	1.000	1.000	0.000	171070	248211	204611			0.01667		
OK	1639	571		S05, W14x34		1.20000	1.20000	1.20000	0.0000	1.000	1.000	0.0000	111468	0.0000			0.00022		
OK	0.890	0.291		A36, W14x34	248211	0.0000	-36.000	0.0000	1.000	1.000	1.000	124106	163028	149207			0.00303		
OK	197	601		C6, W14x170		0.0000	0.0000	0.0000	0.0000	1.000	0.000	41548	122562	43370			0.01061		
OK	0.883	0.157		A36, W14x170	413986	-1399.4	566.636	-76.111	1.000	1.000	0.000	198842	248211	203264			0.02000		
OK	270	602		C6, W14x170		0.0000	0.0000	0.0000	0.0000	1.000	0.000	13097	155027	47005			0.00078		
OK	0.845	0.188		A36, W14x170	413986	-427.55	714.283	-84.034	1.000	1.000	0.000	198842	248211	203264			0.02000		
OK	198	701		C7, W14x80		0.0000	0.0000	0.0000	0.0000	1.000	0.000	39610	109667	88342			0.01034		
OK	0.800	0.146		A36, W14x80	413986	-675.62	256.575	72.0520	1.000	1.000	0.000	190380	248211	209326			0.02000		
OK	394	702		C7, W14x80		0.0000	0.0000	0.0000	0.0000	1.000	0.000	17302	132221	186272			0.00048		
OK	1.088	0.589		A36, W14x80	413986	-308.88	-300.76	-47.880	1.000	1.000	0.000	190380	248211	209326			0.01667		

Steel Code Checking Result Dialog Box

Preview Window

Prop No: 530

4. Checking Results

Slenderness Ratio  
 $L/r = 156.4 < 300.0$  (Node: 1521, LCB: 1)

Axial Stress  
 $f_t/F_t = 0 / 124106 = 0.000 < 1.000$

Bending Stresses  
 $f_{b1}/F_{b1} = 4301 / 163028 = 0.027 < 1.000$   
 $f_{b2}/F_{b2} = 0 / 148827 = 0.000 < 1.000$

Combined Stress (Tension-Bending)  
 $f_{t+b} = f_{b1}/F_{b1} + f_{b2}/F_{b2} = 0.027 < 1.000$

Shear Stresses  
 $f_{v1}/F_{v1} = 0.000 < 1.000$   
 $f_{v2}/F_{v2} = 0.009 < 1.000$

5. Deflection Checking Results  
 $L/360.0 = 0.0167 > 0.0004$  (Node: 1521, LCB: 44, POS: 3.0e, Dir-Z)

Graphical Results

MIDAS/Text Editor [Untitled.acs]

```

( ) Calculate shear stress in local-z direction (fvz),
  ( LCB = 1, POS = J )
  - Applied shear force / Fvz = 1.60 kN,
  - fvz = Fvz / Awz
  - fvz = 99204.536

( ) Check ratio of shear stress (fvz/Fvz),
  - fvz = 992.263
  - Fvz = 99204.536
  - fvz/Fvz = 0.009 < 1.000 -> O.K.

widas Gen - Steel Code Checking | TWN-ASD96 | Gen 2010

[[[+]]] CHECK DEFLECTION
( ) Compute Maximum Deflection.
  - LCB = 44
  - DAF = 1.000 (Deflection Amplification Factor),
  - Position = -3.000e+004 From (midNode: 518),
  - Def_Lim = -3.925e-004 + DAF * -3.925e-004 (Global Z)
  - Def = 0.017e
  - Def < Def_Lim -> O.K !
    
```

Detailed Results