



STRUCTURAL CONCRETE SOFTWARE SYSTEM

# ADAPT-Builder® 20

## ETABS to Builder Conversion Manual

Copyright© October 2020

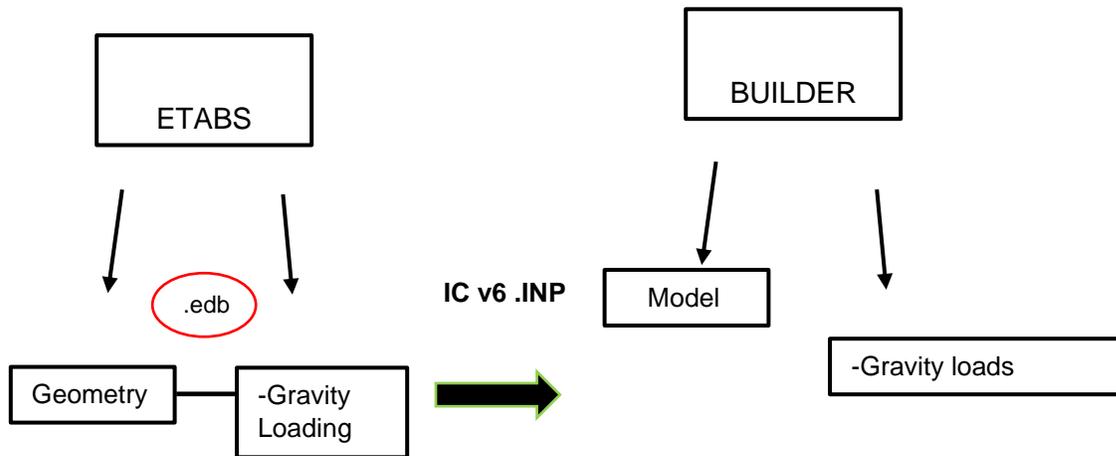


# 1 ETABS to Builder Conversion

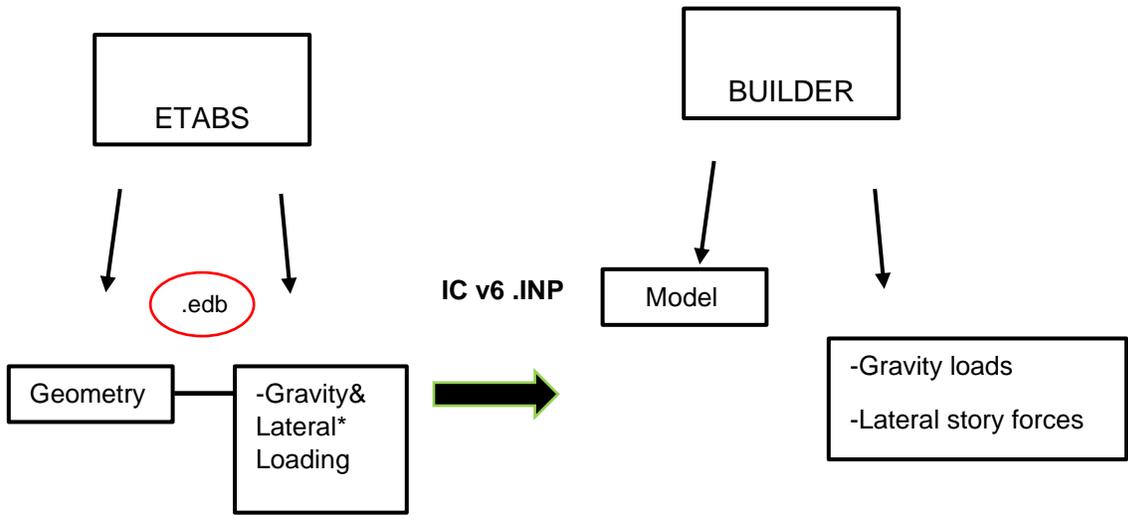
Through use of the **ADAPT-Integration Console v6**, an exchange file is generated including level definitions, component geometry, applied loading, applied lateral story forces and internal component reactions. This file is then imported into Builder for generation of the structural model, including internal reactions and applied forces for all imported load cases that originated in ETABS. An active license is required when generating the .INP file and the ETABS model should be executed with a stored solution.

When imported to Builder, the internal reactions for walls and columns can be directly used for column and wall design within Builder. These reactions can be combined with native Builder reactions, for gravity load cases for example, to obtain a combined combination solution relative to different reaction sources.

In addition, the internal column and wall reactions and applied story forces can be used for single-level slab design incorporating lateral reactions from an ETABS “lateral” model and gravity actions native to Builder. It is also possible to use the imported model, gravity loads and lateral story forces to globally re-run the analysis in Builder to obtain a new analysis solution, relative to the **Edge** analysis engine. To summarize, the following workflows are possible when using the .INP file from the Integration Console:

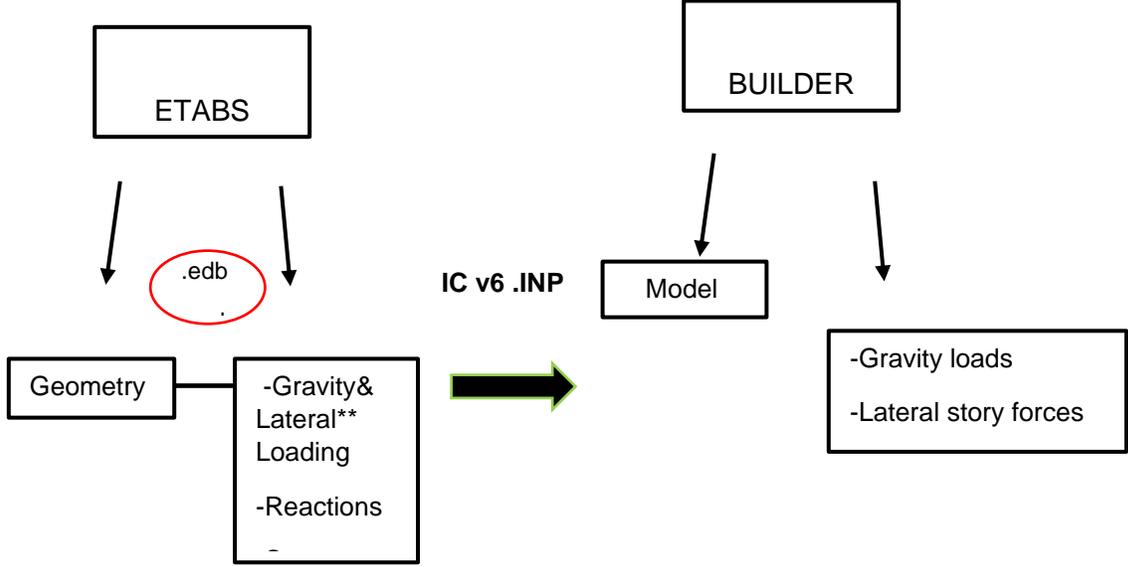


ETABS-Builder Workflow A



**ETABS-Builder Workflow B**

\*Lateral loads are manually input in ETABS



**ETABS-Builder Workflow C**

\*\*Lateral loads are manually or auto input in ETABS

Workflows A and B diagramed above are intended for importing geometry and loading into ADAPT-Builder for a global or single-level model re-analysis with applied loads from ETABS and/or applied in Builder. Also, the column and wall actions can be used directly for column and wall design using ADAPT-Builder and S-Concrete integrated column and wall tools.

Workflow C is the most comprehensive workflow and can be used similar to A and B. However, it also allows a user to re-analyze a floor in single-level mode and include the lateral actions in columns and walls combined with gravity actions present from application of gravity loads in Builder. This workflow includes the presence of internal actions as well as applied story forces, ensuring that equilibrium is achieved for those lateral load cases imported from ETABS.

In the following example, you will learn how to (a) export the .EDB file and create and export the .XML file for ETABS story force data, (b) generate the .INP file using the ADAPT-Integration Console, (c) import the .INP file in Builder for generation of the structural model, applied loads and story forces, and (d) review imported column/wall reactions and applied story forces.

The figure shown below (**FIGURE 1**) is an eight-story reinforced concrete structure which includes columns, shearwalls, and conventional-reinforced concrete (RC) slabs. This model and resulting forces, applied loads, etc. will be exported to ADAPT-Builder.

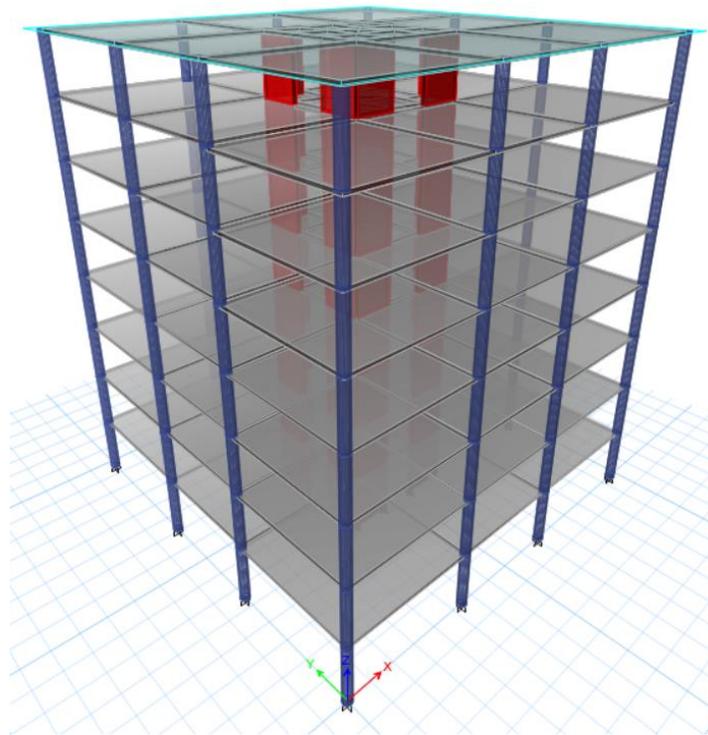


Figure 1 RC Structure as Modeled in ETABS

### 1.1 Preparing the Story Force Data and Exporting .XML and .EDB Files from ETABS

- After the ETABS model has been analyzed and the solution saved, In ETABS, select *File* → *Export* → *ETABS Tables to XML* (**FIGURE 2**). This file contains the applicable story force data relative to lateral loading using the “auto-lateral” function in the ETABS model.

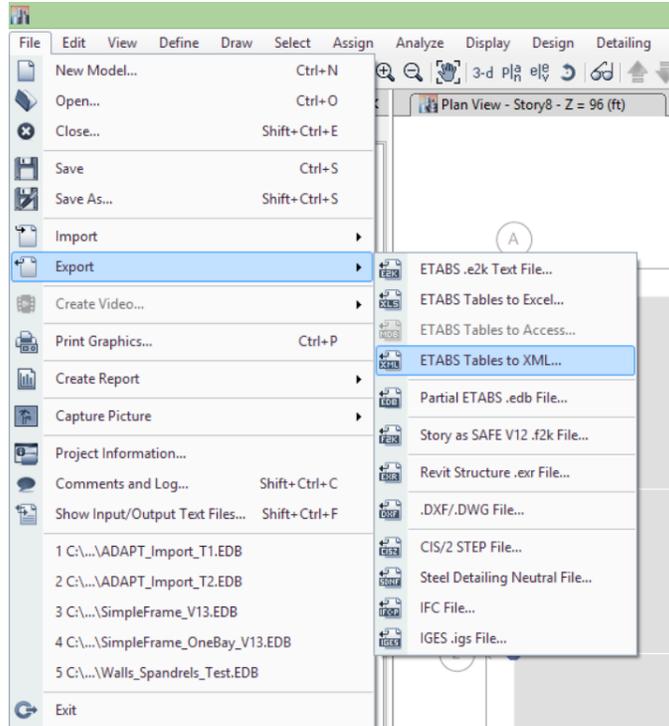


Figure 2 Preparing ETABS .XML File

- In the Choose Tables dialogue box, select Analysis ➔ Results ➔ Structure Results ➔ Story Forces (FIGURE 3).

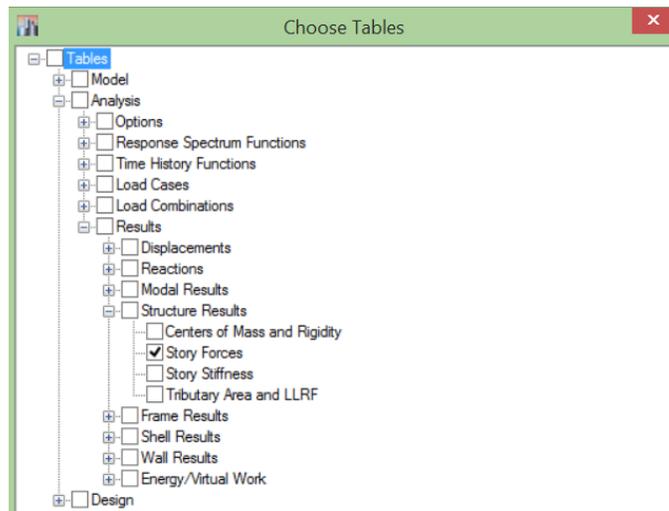


Figure 1 Preparing ETABS .XML File

- In the *Choose Export Units* dialogue box, select the units for *Length*, *Force* and *Temperature* data to be exported from ETABS. For this example, use the default values per US unit selection for the main model as shown in FIGURE 4.

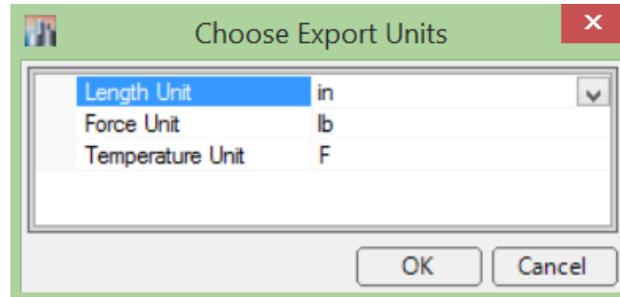


Figure 4 Export Unit Options in ETABS

- In the *Save XML File As* dialogue box, select the location of the .XML file and name the file (FIGURE 5).

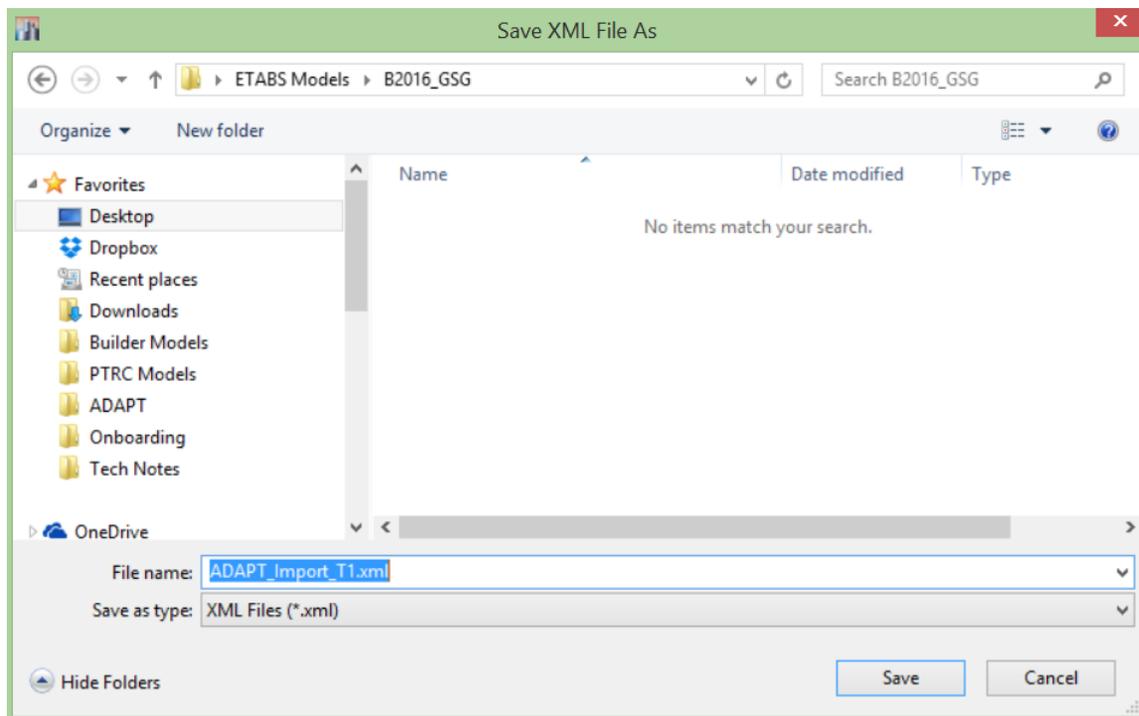


Figure 5 ETABS .XML File Save-As Dialogue Window

## 1.2 Importing the ETABS .EDB and .XML Files into the ADAPT-Integration Console (IC) v6

- Open **ADAPT-Integration Console v6**. Ensure that the **ETABS 2016 (using API)** option is checked in the *Select Program* section (FIGURE 6). Select *Next*.

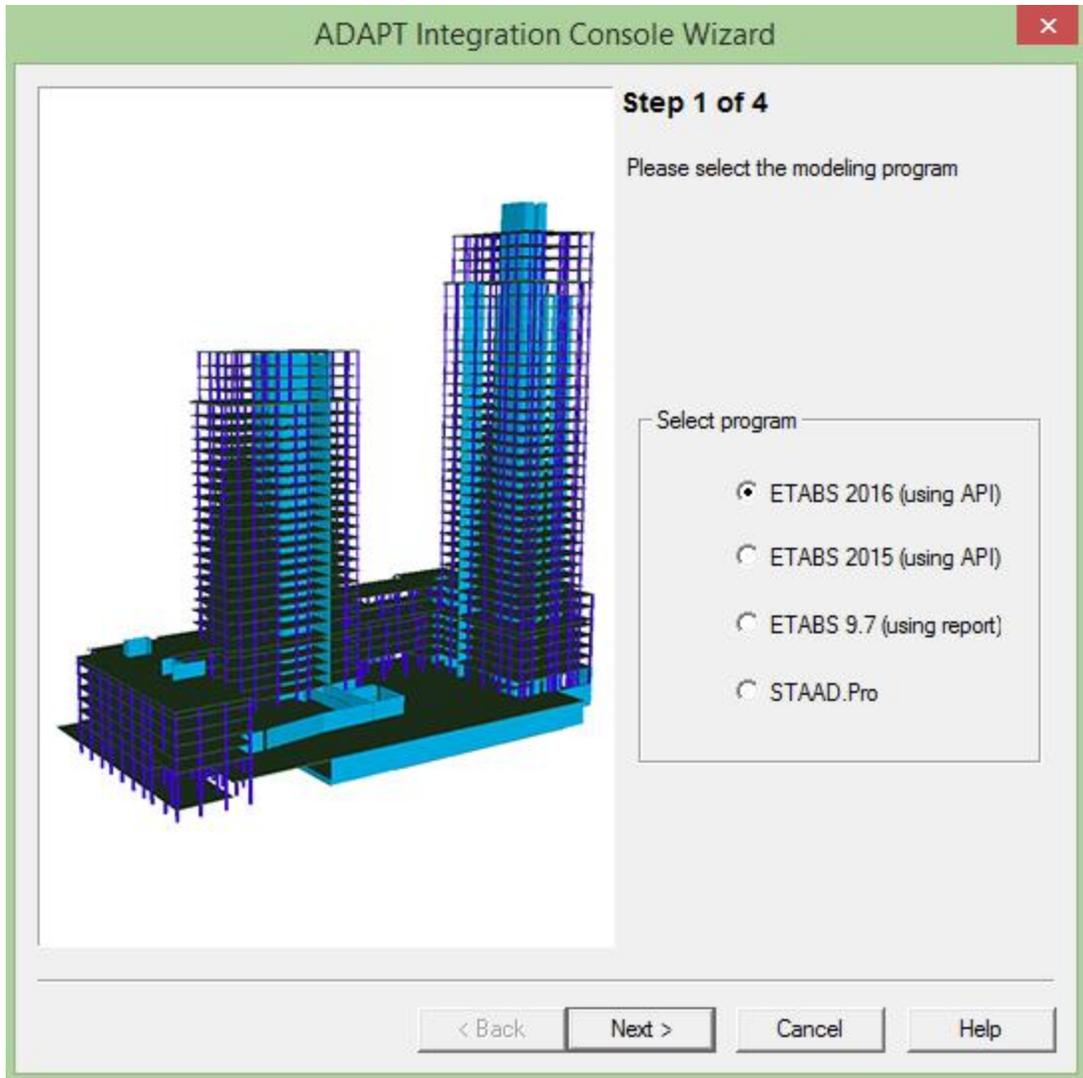


Figure 6 ADAPT Integration Console Wizard

**Note:** In the **ADAPT Integration Console Wizard** dialogue window, the options for *ETABS* and *STAAD Pro* are also available. These options are used for integration with earlier versions of *ETABS* (v9.7 or earlier) and *STAAD Pro*.

- In the *ETABS API Link* dialogue window, browse and select the *ETABS \*.EDB* file and select *Load Model*. See FIGURE 7. Select *Create ADAPT Exchange INP File*.
- After the model has loaded, in the same interface, select *All Levels* for the *Level to Import* and then select the option for selection of the *.XML* file. Browse and select the *ETABS \*.XML* file and select *Create ADAPT Exchange INP File*. See FIGURE 7.

**Note:** An active *ETABS* 2015 or 2016 license is required to access the *ETABS API* and write data to prepare the *.INP ADAPT* exchange file. When this option is selected, *ETABS* will open and close to extract the requested data.

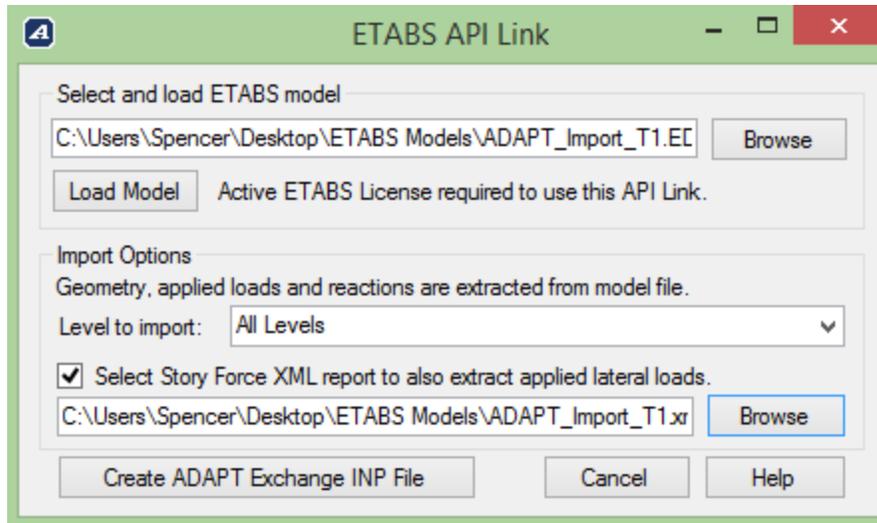


Figure 7 ETABS Link Export Options Dialogue

- After the necessary information is extracted, the message showing *API script completed successfully* will appear as shown in **FIGURE 8**. This message indicates the .INP has been created successfully.

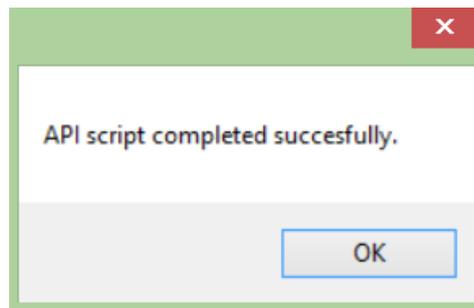


Figure 1 ETABS Link Completion Message

- The generated ADAPT exchange .INP file will be in the same directory as the .EDB file. Close the ADAPT-Integration Console v6. See **FIGURE 9**.

25storyLevel15.INP	1/15/2009 10:09 PM	INP File	111 KB
25storytest.INP	6/15/2011 8:20 PM	INP File	958 KB
ADAPT_Import_T1.INP	7/11/2016 5:25 PM	INP File	549 KB
ADAPT_Import_T2.INP	7/5/2016 2:58 PM	INP File	549 KB
AnishTraining.INP	6/7/2010 8:23 AM	INP File	102 KB

Figure 9 ADAPT Data Exchange File (.INP)

## 1.3 Importing the ADAPT Exchange File into ADAPT-Builder for ETABS

- Open **ADAPT-Builder 20**. Select the appropriate *System of Units* (**FIGURE 10**).



**Figure 10 Builder Platform Module Selection**

**Note:** In the Builder Platform Module Selection screen **Edge** and **Floor Pro** are selected while **MAT** and **SOG** are not selected. The ETABS model import workflow described in this section works when opening Builder in Edge, Floor Pro, MAT, or a combination of modes.

- In Builder, select *File* → *Import* → *INP File*

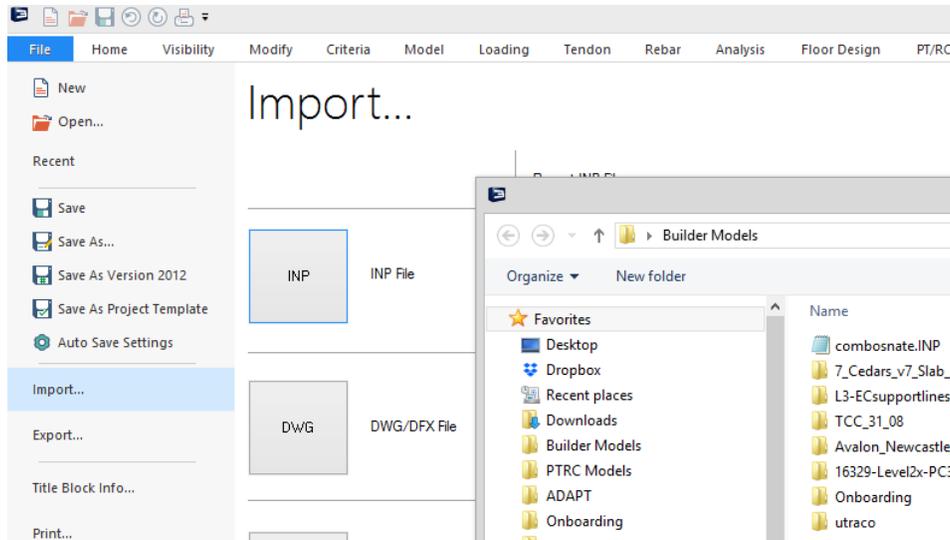


Figure 11 Import ETABS Model into Builder

- In the *Open* dialogue box (FIGURE 12), navigate to and select the ADAPT model exchange file that was created previously.

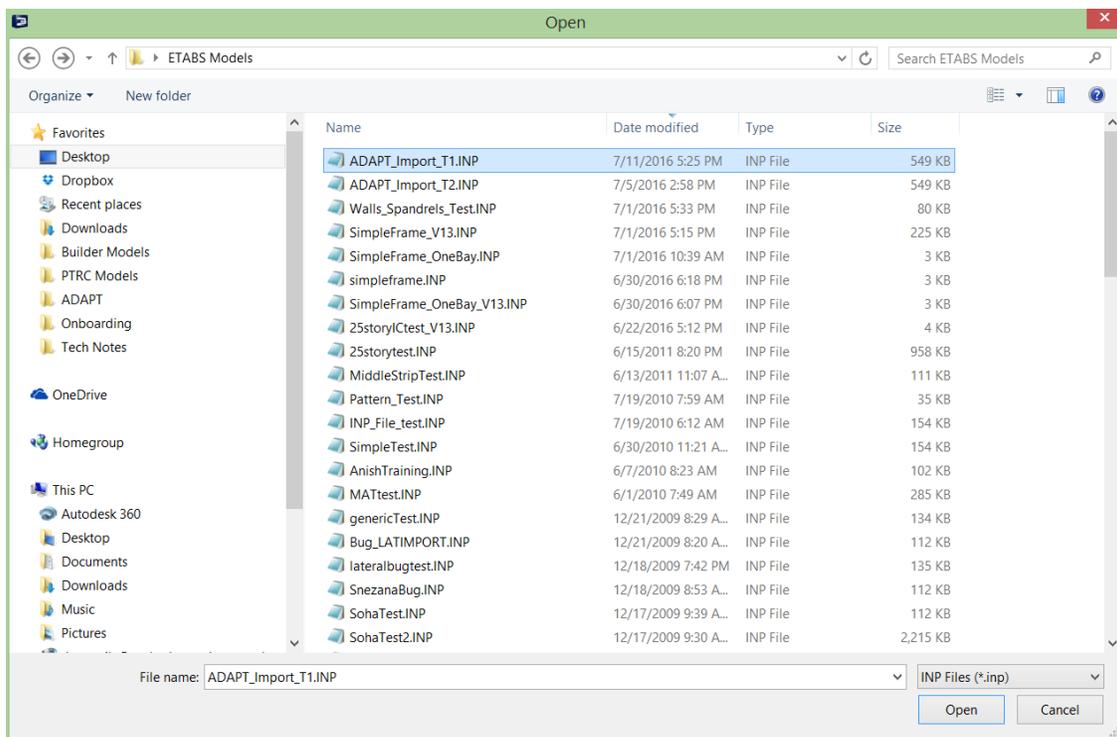
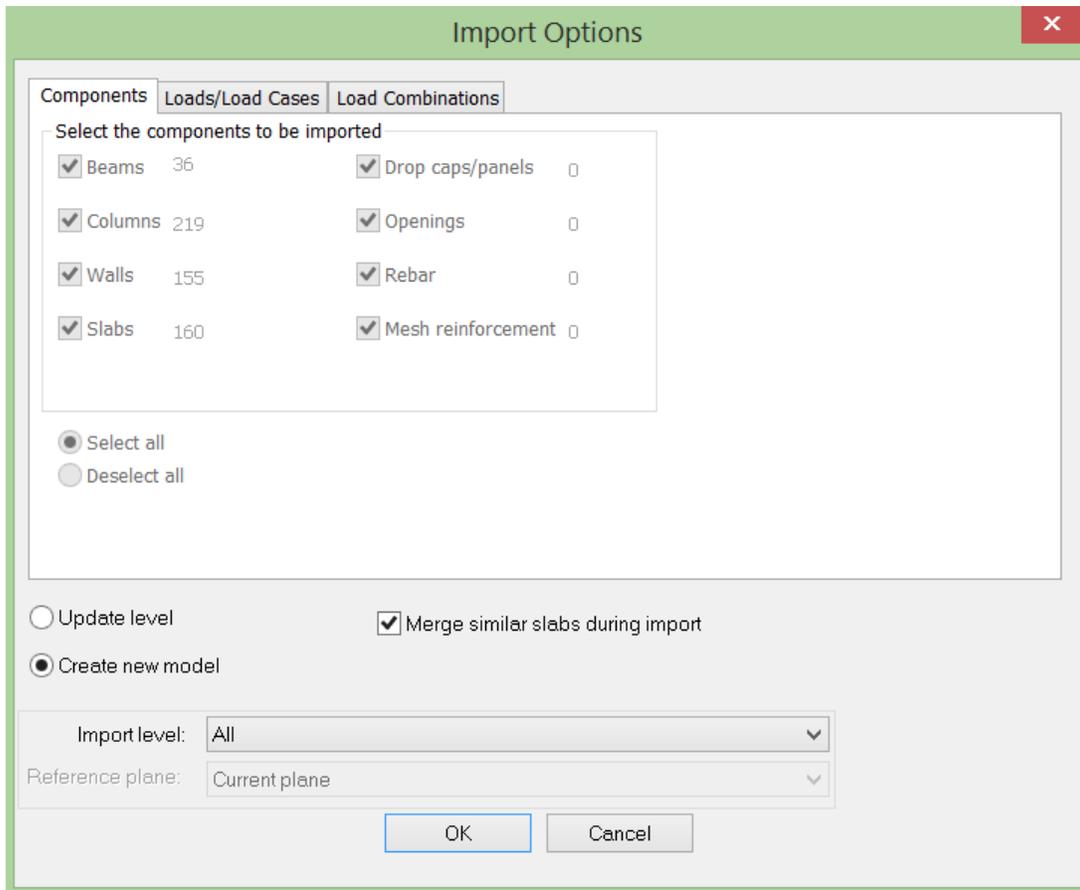


Figure 12 ADAPT Data Exchange File Open Dialogue Window

In the *Import Options* dialogue window you will have the options of which components, loads, load cases and load combinations to import. You can also select to update a model or create a new model, to import the entire structure defined or generate a model of a single level. These can be selected under *Import Level*. By default, the

working reference plane is set to *Current Plane*. When levels defined in the exchange file are imported, the program will update the reference planes. For example, slabs defined at “Level 5” will be assigned a reference plane of “Level 5” instead of *Current Plane*.

The *Import Options* also includes a feature allowing similar slabs to be merged during the import. If slabs that are located on the same plane have the same thickness, offset and properties, the program will merge the slabs into one region.

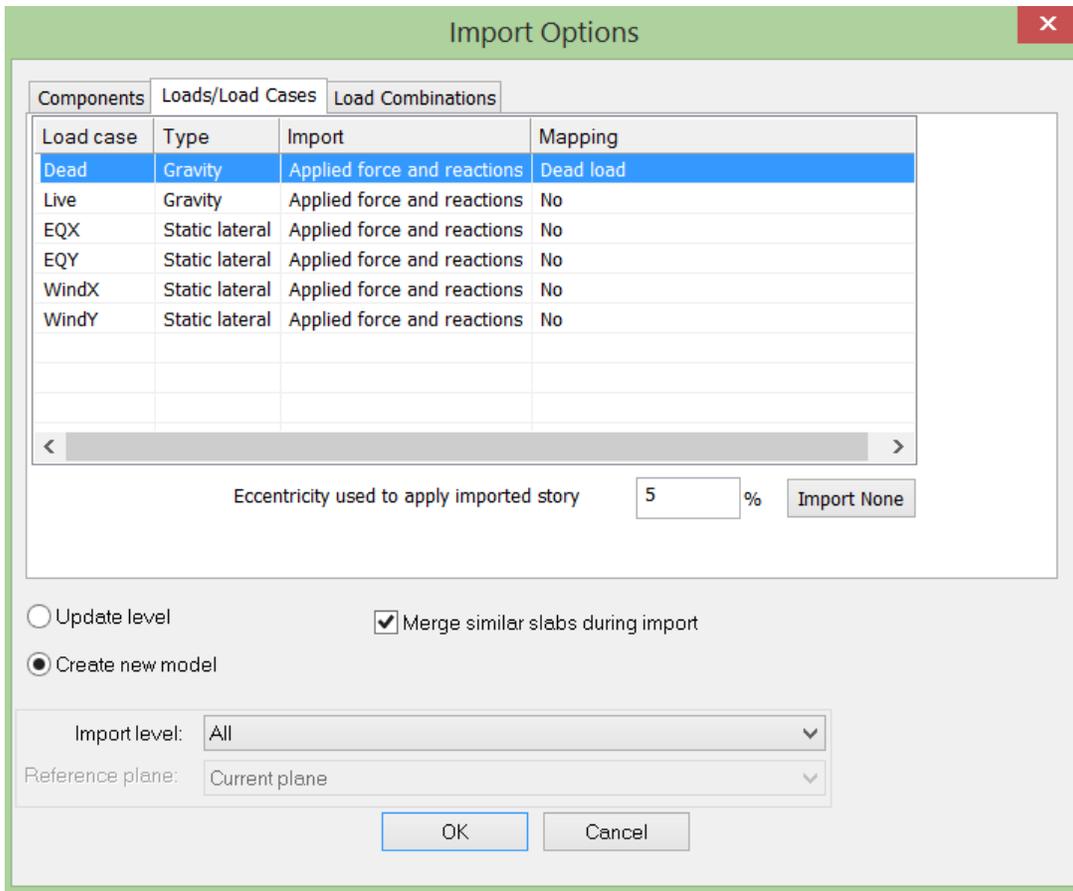


**Figure 13 Import Options Dialogue Window**

The *Loads/Load Cases* dialogue window (**FIGURE 14**) reports the load cases being exported from ETABS and imported to Builder. Each load case is associated with a *Type* (e.g. “Gravity” and “Static Lateral”). These load types are provided for information only and are not treated differently within Builder once imported. The *Import* column reports whether Applied force and/or reactions are included in the import. The *Mapping* option allows load cases exported from ETABS to be mapped to a defined load case within Builder. All applied forces and reactions for an imported load case will be reassigned to the mapped load case.

The option for *Eccentricity used to apply imported story* is used to define the eccentricity % of length for the edge of slab perpendicular to the applied story force. If the model is

re-analyzed in global building mode with the story forces imported from ETABS, the program will consider the direct, entered story force plus any torsional effects derived from the input eccentricity.



**Figure 14 Import Options – Loads/Load Cases**

The *Load Combinations* dialogue window (**FIGURE 15**) reports all load combinations defined in ETABS. The combination components (load cases) and load factors are listed under the *Details* column. The user also has the option on whether or not to import the load combinations, as combinations are commonly created and defined within Builder.

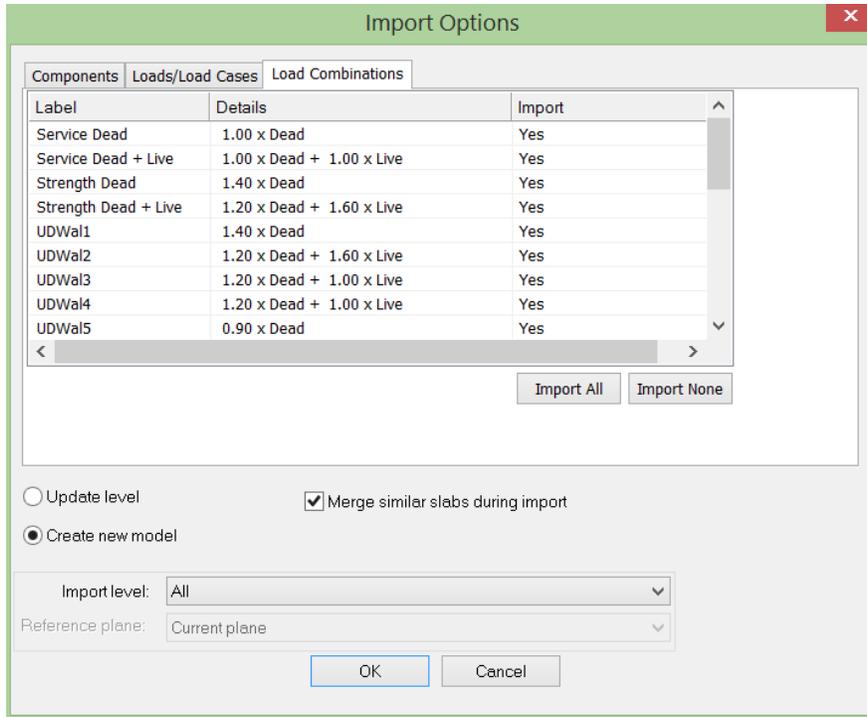


Figure 15 Import Options – Load Combinations

When the import is complete, the structure will open in *Multi-Level Mode*  mode. A plan view will appear as shown in **FIGURE 16**. All imported levels, components, and loads will be shown.

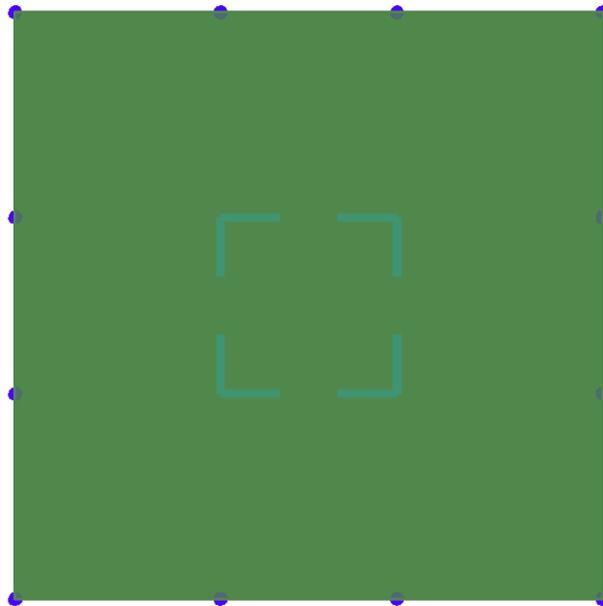


Figure 16 Imported 8-Story Structure Shown in Plan View in ADAPT-Edge

- Click on the *Top-Front-Right-View*  from *Visibility* → *Camera/Zoom* and you will see an isometric view of the imported structure in **ADAPT-Edge** (**FIGURE 17**).

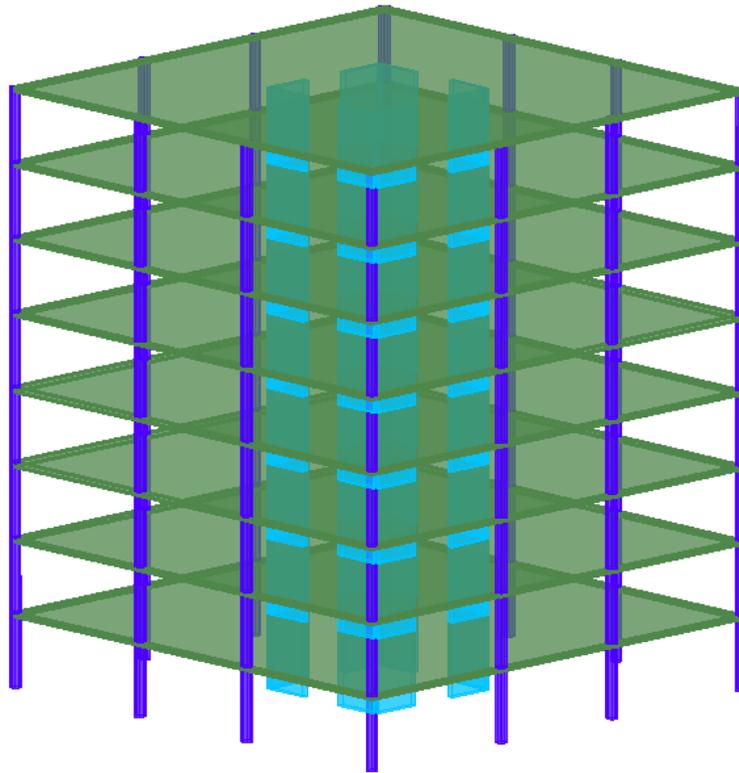


Figure 17 Imported Multistory Structure Shown in 3D Isometric Wireframe View in ADAPT-Edge

#### 1.4 Imported Applied Loads and Reactions

Upon completion of importing the ETABS model into Builder, there are options available to review applied loads (gravity and lateral story forces) and column and wall reactions. If both the .EDB and .XML files have been used for the import of data, the user can re-analyze a single-level with consideration of the imported reactions. Imported story forces can be used for the analysis of a global model.

- To review the applied gravity loads, go to *Visibility* → *View Settings* . Select the *Loads* tab to choose which loads are to be viewed (Error! Reference source not found. **FIGURE 18**).
- In the main user interface, the loads will appear as shown in **FIGURE 19**. Note the loads selected are those imported from ETABS. If other loads were defined within Builder, they also would be available for selection.

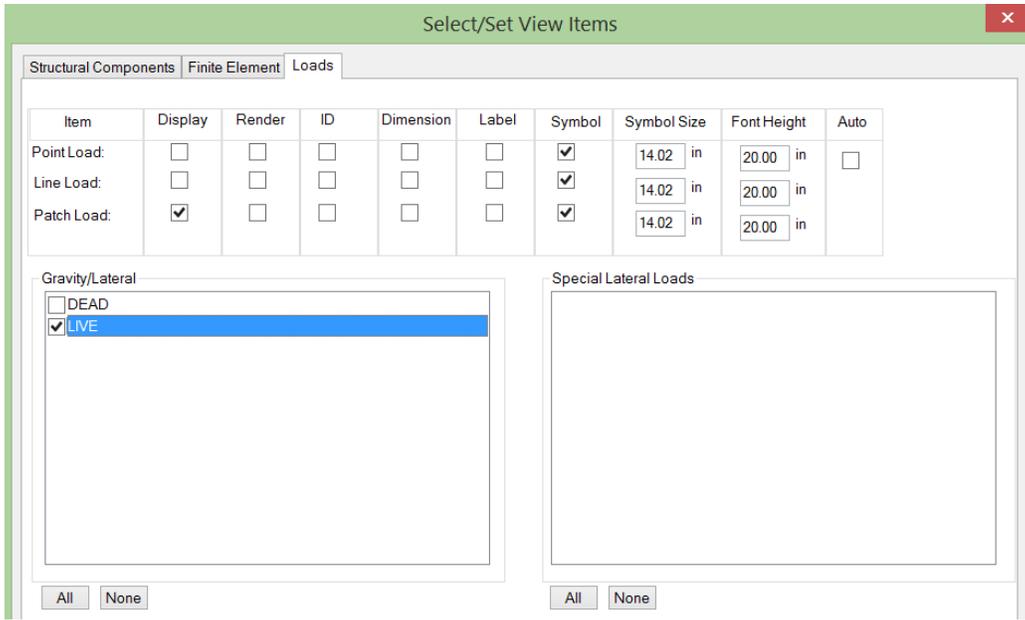


Figure 18 Select/Set View Items - Loads

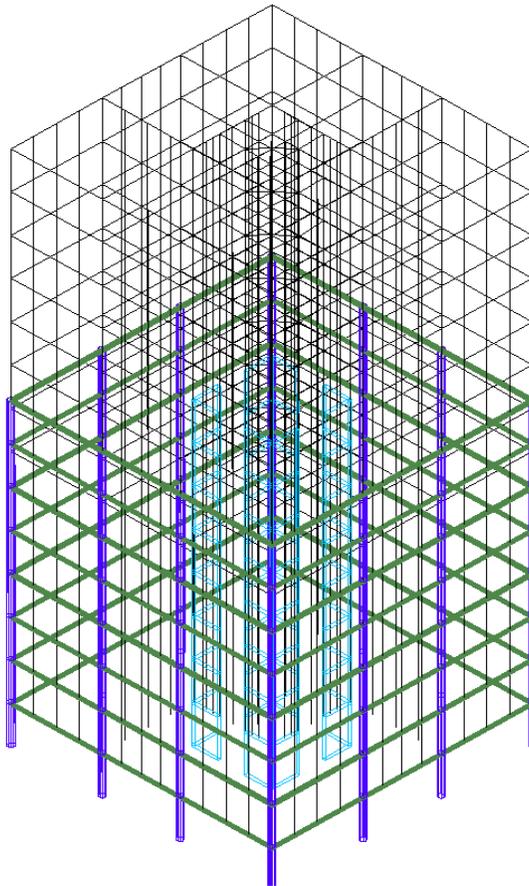
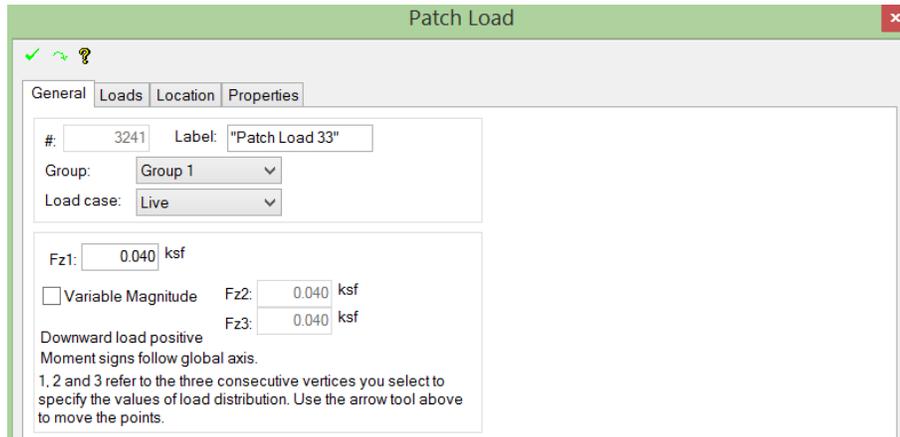


Figure 19 Builder Model with Imported ETABS Gravity Loads

- Double-click on any of the patch loads graphically represented in the model. This will open the *Patch Load* dialogue window. Here the user can review information related to the patch load imported from ETABS. See **FIGURE 20**. Note that any load imported from ETABS will include "" as part of the default load label.



**Figure 20 Patch Load Dialogue Window**

- To review the applied lateral story forces, select *Loading* → *Lateral Load* → *Lateral Load Wizard*. The *Lateral Load Wizard* reports load cases and story forces at reference planes assigned to the load case imported from ETABS. See **FIGURE 21**.

If the option for *Update existing load case* is selected, the user can revise options related to the selected case. These include the *Force* applied at each *Reference Plane*, the applied loading *Direction*, and *Eccentricity*. If a global model is analyzed with use of the imported ETABS story forces, the program will utilize the Seismic mass source combination, as shown below the story force table, for rationing the total story force to nodal forces and moments at shell nodes.

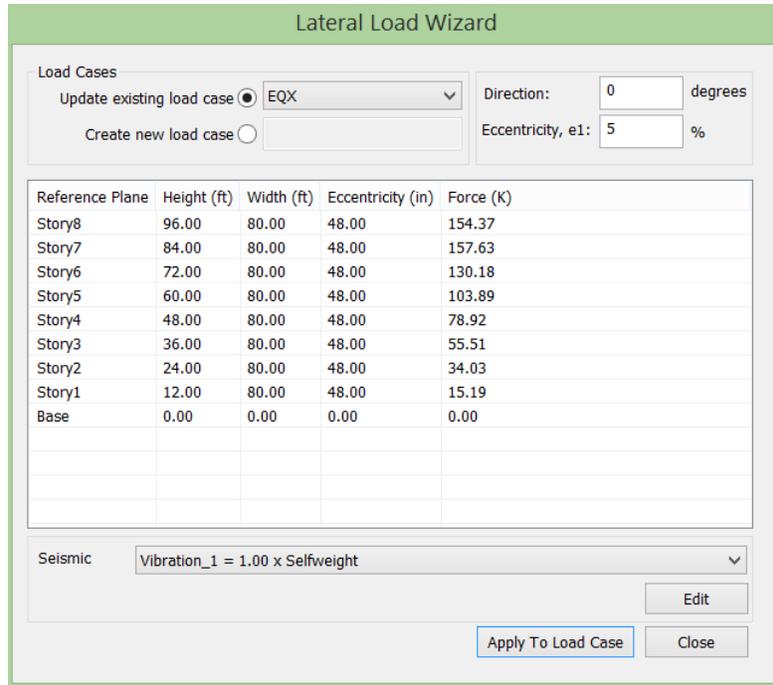


Figure 21 Lateral Load Wizard

- Column and wall actions that are imported from ETABS, be it gravity or lateral, can be viewed graphically using the *Result Display Settings* tool  to launch the graphical results browser. For the review of graphical results, it is recommended to use this tool in *Single-Level* mode for this example.

From the *Case* drop-down menu, the program includes the option for selecting any imported (ETABS) or native (BUILDER) solution set for the available load cases. In this example, the Builder model has not been analyzed, so only the imported ETABS load cases are reported. This is denoted by the *(ETABS: G)* as shown in **FIGURE 21**. The “G” denotes that the load case solution is from a global analysis run. If “L” is shown, the solution is from a single-level analysis run.

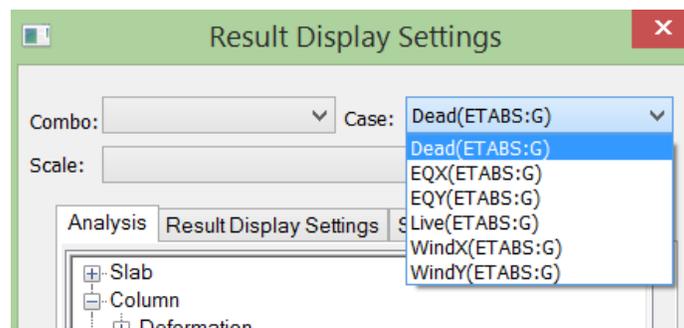
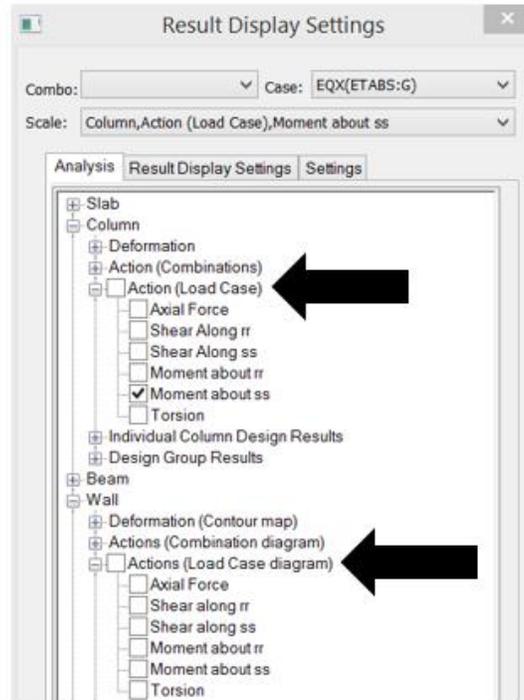


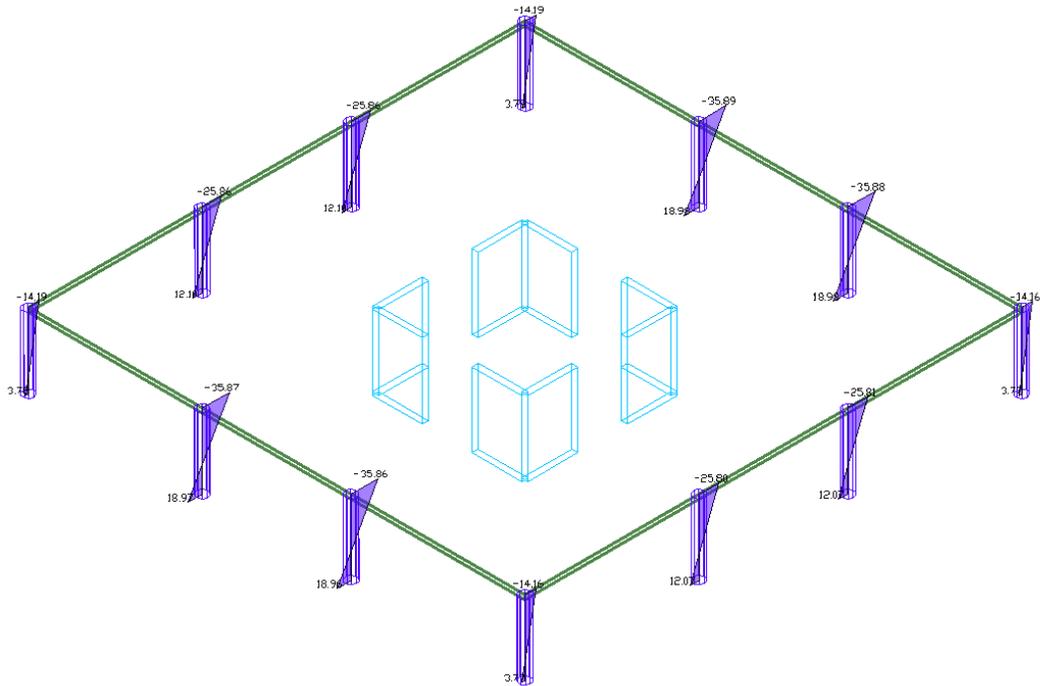
Figure 22 Result Display Settings – Load Case Selection

In the Result Display Settings browser, the Column and Wall branches include the options for displaying Actions (Load Cases). See FIGURE 23.



**Figure 23 Result Display Settings – Column and Wall Case Action Selection**

From the load case pull-down menu, select *EQX (ETABS):G* to view internal column reactions for this example. Select *Column* → *Action (Load Cas)* → *Moment About ss*. **FIGURE 24** shows the resulting graphical results.



**Figure 24 Column Moments for Imported ETABS “EQX” Load Case**

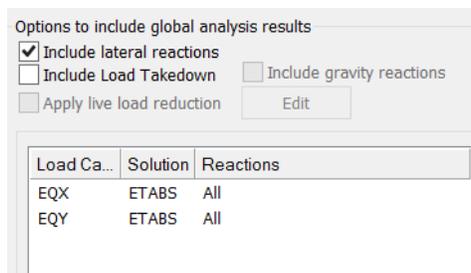
- Imported reactions for columns and walls can be combined with loads applied within Builder for single-level analysis. This condition might be applicable for the design of a slab and/or beam system with combined actions from different sources. For example, the lateral actions based on an ETABS analysis and the gravity actions based on a Builder analysis.

Select *Analysis* → *Analysis* → *Execute Analysis*. **FIGURE 25** shows the *Analysis Options* dialogue screen. The following descriptions are given for use of options within this screen that are new and/or applicable to re-analysis incorporating reactions imported from ETABS:

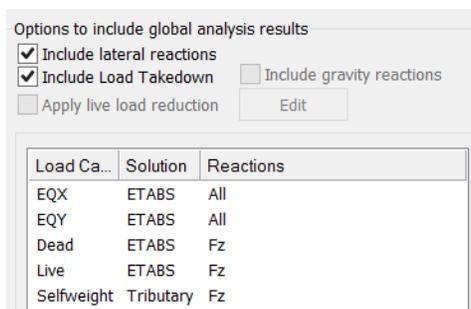
1. **Select load combinations for analysis** – Select the combinations to be used in the analysis of the model in single-level mode. Combinations selected that include load cases imported from ETABS will activate option (3), allowing the user to select to include lateral reactions and load takedown of gravity reactions.
2. **Options to include global analysis results** – This option allows the user to include saved reactions (from ETABS or a native Builder global solution) in the analysis of a single-level. In the context of analyzing a single-level with column and wall reactions imported from ETABS, this option would be used if the user is combining a lateral solution with a

gravity solution for the purpose of designing slab or beam sections within Builder. There are multiple options available to select.

- a. **Include lateral reactions** – When this option is selected, the program will display all available global or imported lateral load case solutions in the window to right in FIGURE 26. The program will itemize each *Load Case*, associated *Solution* source and which *Reactions* are included. In this example, **EQX** and **EQY** are reported with the solution source being **ETABS**. Note that the load case list is dependent on which load combinations are selected to be run for the single-level analysis. If the load combination selected set does not include a load case from a global solution, the list shown below would be blank.



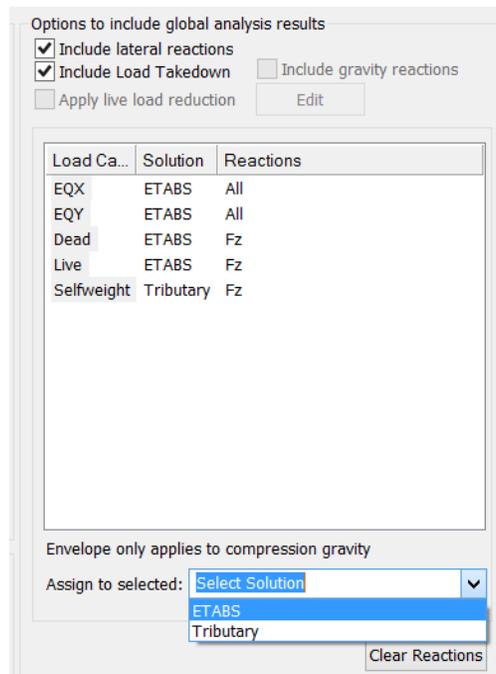
- b. **Include Load Takedown** – When this option is selected, the program will display all available global or imported gravity load case solutions in the window to the right shown in FIGURE 26. In other words, this is the setting that controls the application of axial load takedown to the single-level being analyzed. In this example, **Dead** and **Live** are reported with the solution source being **ETABS**. Note that the **Selfweight** load case is derived from the **Tributary** solution. This is a Builder solution from the Tributary Load takedown feature included in the previous release.



- c. **Include gravity reactions** – This option is only available when a model is opened in ADAPT-MAT. The intent of the option is to include not only the axial load take-down as described in (b), but to include bending effects due to gravity loading also. The solution

source for this option can be an imported solution or a Builder solution.

3. **Assign to selected** – If multiple solutions are available for a specific load case to reference for a single-level analysis run, the user can select load cases in the window to right, shown below in FIGURE 25, and from the *Assign to selected* pull-down list, select which solution to assign to the selected load case. Note that when *Envelope* is listed as an option in the pull-down list, the program uses the enveloped values for all load cases when applying solution reactions to the single-analysis run.



**Figure 1-252 Options to Include Global Analysis Results**

The option for *Clear Reactions* will launch the *Reaction Manager* window. This option can also be launched from *Analysis* → *Reactions* → *Stored Reactions*. . See **FIGURE 27**. This dialogue includes options for managing saved solutions, native or imported. Solutions that have been solved natively within Builder or imported from the ADAPT-Integration Console, will be stored and solutions available for display and/or selection for a single-level analysis until the solutions are deleted using this tool.

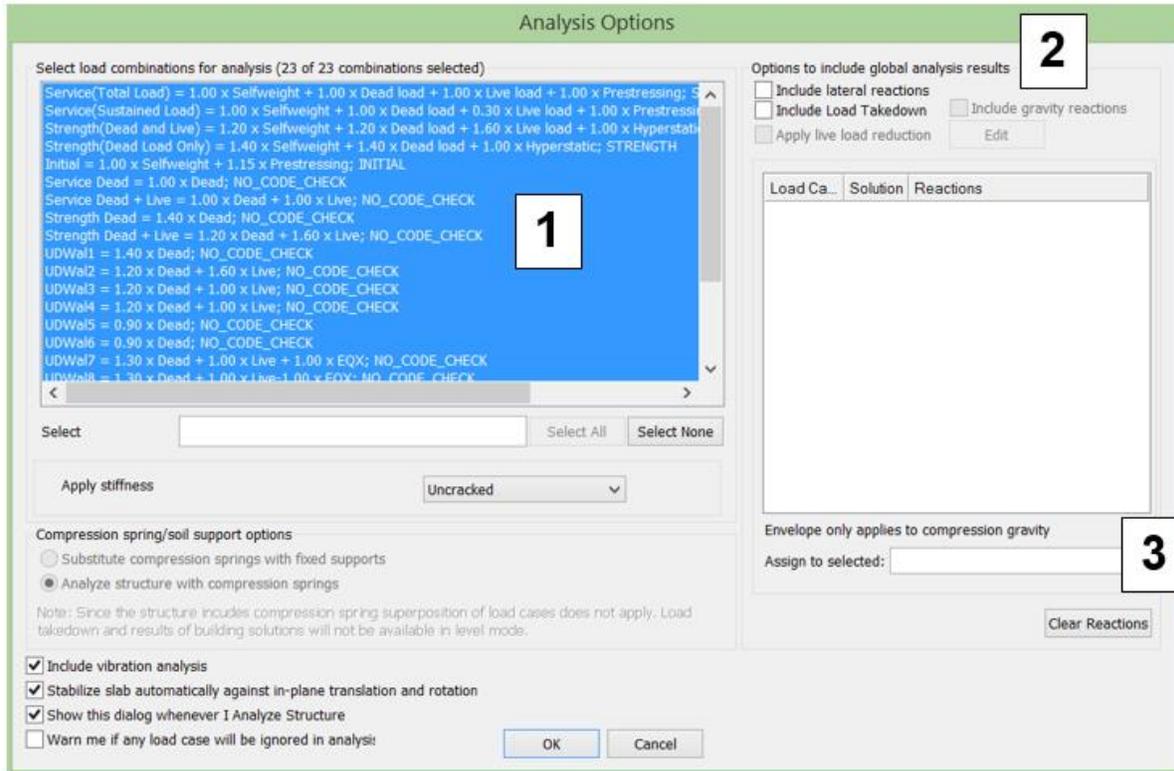


Figure 26 Analysis Options

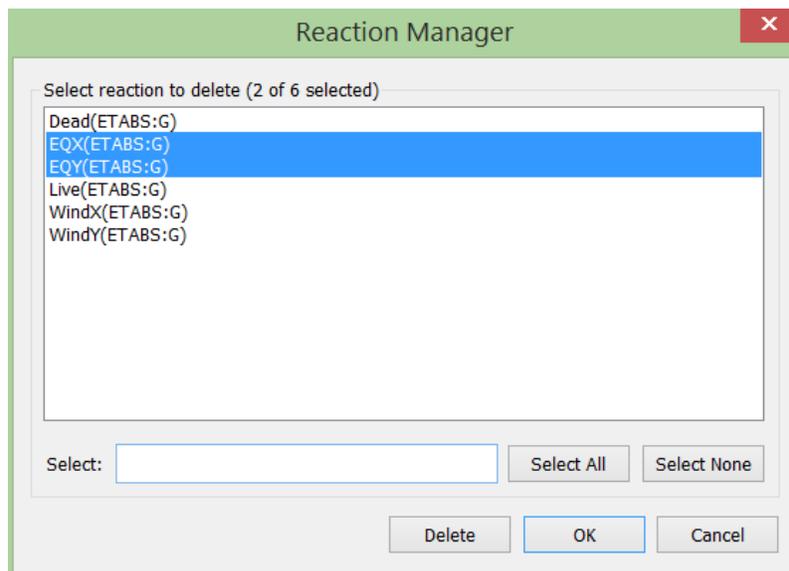


Figure 27 Reaction Manager