VisualPlate 6.0

User's Guide

Updated: 1/29/25

VisualPlate 6.0 User's Guide

Table of Contents

1 Help Topics

1.1 Welcome to VisualPlate 6.0

A New Commercial Version is Available.

IES has upgraded VisualPlate 6.0. The latest release can be found on our webstie at: www.iesweb.com/downloads

VisualPlate will help you solve a wide variety plate bending problems using any material. This powerful analysis program is designed to be flexible so that you can efficiently solve your unique problem. It automates much of the work involved in finite element modeling that can be tedious in a more general tool. The software will analyze your model to determine stresses, moments, shears, and displacements using finite element analysis.

Getting Started

- Use **File | Open Example Project** to see sample projects.
- Feature List
- Program Layout
- Upgrade Guide (what's new)
- FAQ Answers at iesweb.com for business, licensing, installation issues.

Help Notation

Menu items appear like this: **File | New**.

Keystrokes or mouse commands appear like this: ***Shift+Click***.

Disclaimer

VisualPlate is a proprietary computer program of Integrated Engineering Software (IES, Inc.) of Bozeman, MT. This product is intended for use by licensed, practicing engineers who are educated in structural engineering, students in this field, and related professionals (e.g. Architects, Building Inspectors, Mechanical Engineers, etc.). Although every effort has been made to ensure the accuracy of this program and its documentation, IES, Inc. does not accept responsibility for any mistake, error, or misrepresentation in, or as a result of, the usage of this program and its documentation. (Though we will make every effort to ensure that problems that we can correct are dealt with promptly.) The results obtained from the use of this program should not be substituted for sound engineering judgment.

License and Copy Restrictions

By installing VisualPlate on your computer, you become a registered user of the software. The VisualPlate program is the copyrighted property of IES, Inc. and is provided for the exclusive use of each licensee. You may copy the program for backup purposes and you may install it on any computer allowed in the license agreement. Distributing the program to coworkers, friends, or duplicating it for other distribution violates the copyright laws of the United States. Future enhancements and technical support depend on your cooperation in this regard. Additional licenses and/or copies of VisualPlate may be purchased directly from IES, Inc.

IES, Inc.

Integrated Engineering Software, Inc.
3740 Equestrian Ln Unit 1
Bozeman, MT 59718

Sales or Licensing: 406-586-8988, sales@iesweb.com
Technical Support: support@iesweb.com

1.2 Key Features

Modeling

- Quick-start with typical geometries
- Create any plate geometry by sketching boundaries
- Import from CAD for complex and detailed boundaries
- Work with column-lines to organize and modify structures
- Create plates with various thicknesses
- Define line stiffeners (e.g. beams) with various shapes and material properties
- Define point supports (e.g. columns)
- Define line supports (e.g. walls)
- Easily refine finite element mesh
- Create new model objects by copying existing items

Loading

- Model objects can be loaded in multiple service load cases (e.g. Dead, Live, etc.)
- Automatic building code load combinations are available
- Includes IBC, ASCE, and NBC Load Combinations
- Customizable building code combinations (see Load Case Manager)
- Create custom load combinations in any project
- Apply point loads, line/distributed loads, and area load pressures to the structure
- Copy and paste loads to objects
- Copy and scale loads to other load cases

Analysis

- FEA model is constructed automatically by the software
- Automated "background" analysis is fast
- Plate elements use a "thick-plate" formulation for accurate shear results
- Advanced error-checking and reporting
- Export FEA model to VisualAnalysis to better understand model details or to perform a more sophisticated analysis

Reporting

- Quick Full Report includes graphics and all details, with options
- Custom reporting to include just the information you need
- Print Preview mode while working with reports
- Paste any graphics into your report
- Customizable page margins, fonts, colors
- Use your own company logo in report page headers
- Print to any printer including PDF

- Export to text clipboard or save to other formats like .xlsx

General

- 3D Graphics
- Simple, standard Windows interface for easy navigation
- Unlimited Undo & Redo commands
- Work in any unit system, perform math on input, use custom unit 'styles'
- Program is self-documenting with tooltips on commands and input parameters
- Numerous preference settings for better defaults
- Free training videos provided for learning efficiency
- Free technical support email with fast, friendly turnaround

Limitations

- Does not perform member design or check design specifications
- Does not produce structural drawings
- Cannot model a system of disconnected plates.

Be a Squeaky Wheel

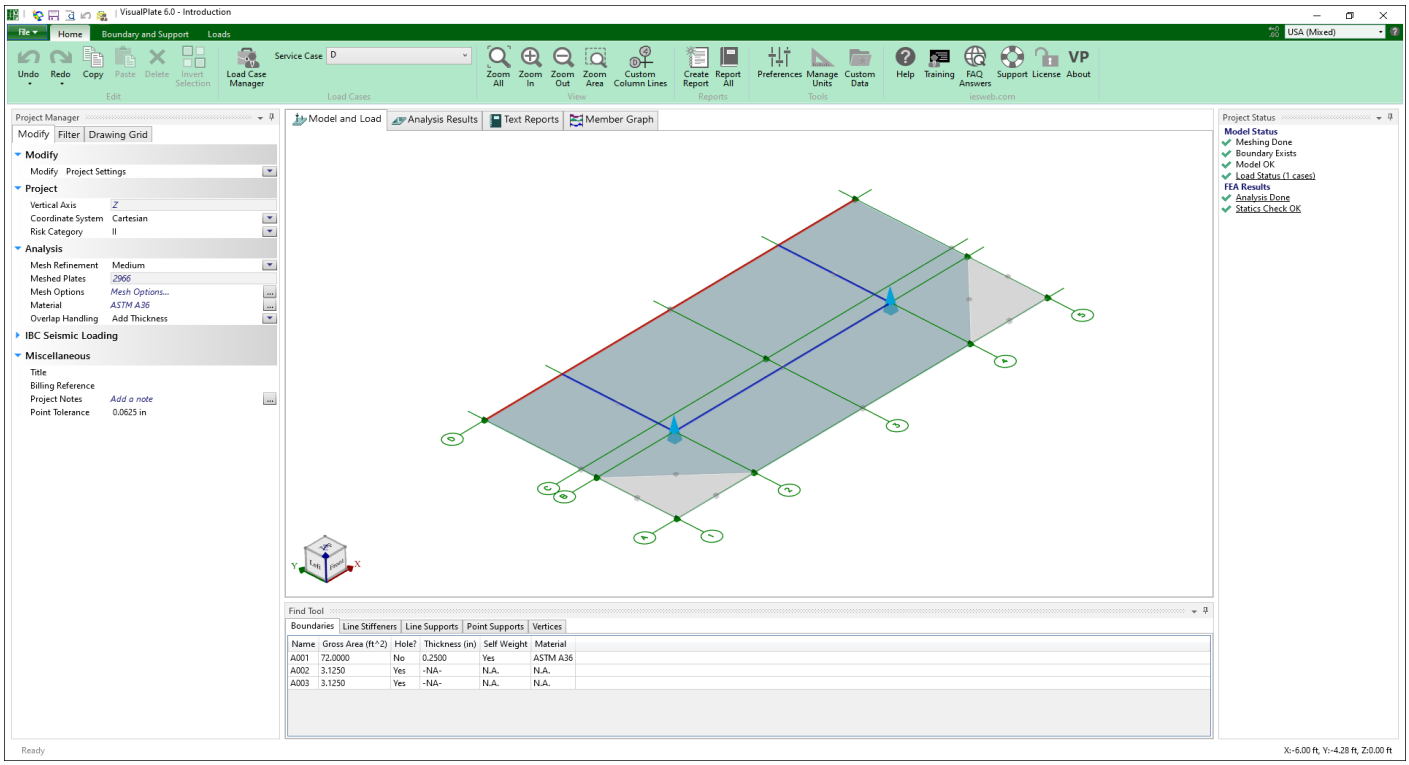
If you need a new feature, please let us know. We are always looking for ways to improve products in ways that you desire. See Technical Support.

1.3 Program Layout

The best way to learn VisualPlate is to use and explore the program to get to know what is available under each button or menu. Several Tutorial Videos are also available which explain many features of the software.

Screen Layout

The image below introduces the program terminology used in this help file and the training videos. Panels may be resized by dragging their dividers or repositioned by dragging their title bars or right-clicking on the title. Use the "pushpin" icon to collapse panels temporarily to gain more space for working. Hold the mouse pointer over the screen image below for information about each area of the program.



Main Menu / Toolbar

The main menu, Toolbar, or Ribbon, contains various commands to direct VisualPlate. Each is organized within a group to help locate them quickly. Each command has a description which appears when the mouse pointer is hovered over it. Many have hot-key shortcuts.

Project Manager

The Project Manager provides immediate access to frequent operations in VisualPlate. This tool is docked on the left side of the window by default and displays various tabs depending on the active window. This window can be docked on the left or made to float independently if more space is needed to work. Alternatively, drag the side border to make it wider or narrower.

- The **Modify** tabs are used to change the project settings or the properties of selected objects in the Model and Load View.
- The **Filter** tab is used to control what is shown or hidden in the active view.
- The **Drawing Grid** tab is used to control the Sketch Grid to aid in drawing models in the Model and Load View.
- The **Result** tab replaces the Modify tab when the Analysis Result View is active. This tab provides key result information for the active load case.
- The **Tables** tab is used to add available tables to the report.
- The **Report Filter** tab is used to define the report settings, apply the model filters, modify the parameters of the table selected in the report, and select which table columns to display.

Graphic Views

These views provide a way to view the model, analysis results, and reports. Each tab displays different options and will provide different information in the Project Manager and Find Tool. Some Graphic tabs will only appear based on objects in your model, such as the Member Graphs.

Project Status

This panel provides a quick update on what is done, what is in-progress, and whether things are working or failing in your model or checks. Click on any item that is underlined for more information, a report, or a dialog containing quick actions.

Pipeline Status

Shows background meshing and analysis progress. Background processing is done on a separate thread of your processor so you may continue working while the program runs. The only time you need to wait for the program is when the mouse cursor changes into an hour-glass or if you wish to view the analysis results that are currently in-progress. Detailed progress bars are available for background activity by clicking on the status-bar at the bottom of the screen.

Find Tool

The Find Tool provides an efficient way to view, select, and edit boundaries, supports, loads, etc. This tool is docked on the bottom of the window by default. Use ***F7*** or the push-pin icon to auto-hide this panel. When docked, drag the side border to make the panel larger or smaller. The Find tool allows you to find, select, edit, and delete objects even if they are not visible in the active window. ***Double-click*** on an element (boundary, load point, point support, etc) and the graphics window will zoom-in to show that element, if it is visible. Lists shown in the Find tool can be sorted by clicking on a column header (***click*** again to reverse the order). Select items just like any list in Windows using the ***Shift*** and ***Ctrl*** keys to select a range or to toggle individual items.

Units & Precision

Above the toolbar on the far right is the Units drop-down for selecting the way physical quantities are displayed. Change the number of decimal places or significant digits using the icon to the left of the unit selector. Go to **Home | Manage Units** to create custom unit styles or edit existing unit styles.

Data Entry: Physical Quantities

Enter values in any unit style. Enter any number or math expressions followed by a known abbreviation. Length units may be entered in "ft-in-16ths" notation as well. Entered values are converted and then redisplayed in the current 'display' units.

Mouse and Keyboard Commands

Selection:

- ***Click*** to select (mouse hover indicates what object will be selected)
- ***Click*** in the 'whitespace' of a view to unselect everything and access Project Settings
- ***Ctrl+Click*** to toggle object selection without affecting other objects
- ***Shift+Click*** to select all objects of a given type
- ***Shift+Drag*** draw a selection box (left-to right selects fully enclosed objects, right-to-left selects any partially enclosed objects)
- ***Shift+Ctrl+Click*** to select items of one type with the same Name Prefix as the item clicked on

Zoom:

- ***Scroll Mouse Wheel*** with the pointer over the point to zoom in or out
- ***Double Click Mouse Wheel*** to Zoom All

- ***Ctrl+*** (plus) and ***Ctrl-*** (minus) keys
- ***Ctrl+Home*** for zoom all/extents
- ***Ctrl+End*** to enable the **Home | Zoom Area** command then ***Drag*** to create the Area

Pan:

- ***Drag Mouse Wheel*** to pan
- ***Shift+Arrow*** keys will also pan

Rotate:

- ***Ctrl+Drag Mouse Wheel*** to rotate the view
- ***Click*** on a face, edge or corner of the Cube in the lower-left corner of the graphics to rotate the view
- ***Ctrl+Arrow*** keys will also rotate

Context Menu:

- ***Right-Click*** the mouse for a short menu of relevant commands based on the view and what is selected
- ***Shift+F10*** also display the context menu

Hot Keys:

- ***Alt*** will expose the hot-keys in the main menu
- ***F1*** Help
- ***F7*** Show or hide the Find Tool
- **Esc** Cancel the Graphic drawing and enter the Draw Nothing mode
- **Delete** the Graphic selection
- ***Ctrl+C*** Copy graphic image to clipboard
- ***Ctrl+V*** Generate copies, or paste graphics in Report View

Miscellaneous:

- ***Drag*** in the Model View to sketch boundaries, line supports, line stiffeners, etc.
- ***Double Clicking*** in the Analysis Result will generate a Text Report for the object. Double-clicking on an element or node in the Find Tool will Zoom to that item.

Context Menu:

Right-Click the mouse for a short menu of relevant commands based on the view and what is selected.

Middle-Mouse "Button" in Windows

Depending on your system, you may need to go into Control Panel, Hardware, Mouse, and set the wheel button to behave like a "middle button click". Some mouse utility programs may override that setting or it may not be set up on some versions of Windows.

1.4 Upgrade Guide

Version 6.0 (April 2022)

Watch the **Upgrade Guide Video** to see the new features of VisualPlate 6.0 in action.

New Features

- Boundary's mesh can be refined at load point and point support locations
- Improved handling of models with misaligned geometry
- Improved drawing grids
- Improved the performance of generated loads
- Program ensures area loads are properly modeled before starting the analysis
- Line stiffener loads remain constant when switching between resultant and distributed
- Load Case Manager columns now retain their size and order
- Plate result diagram overhaul
- Improved filtering of table extremes
- Preferences created for justification of text and data in reports
- Individual column justification added in reports (right click on column)

Fixes & Minor Changes

- Improved Help File documentation
- Load Area information added to Area Uniform Loads tab in the Find Tool
- Added Sigma Top columns to the Plate Stresses table
- Crash recovery file improvements
- Updated the c++ runtimes to the current standards

1.5 Release History

Prior Releases

- Version 5.0 (November 2018)
- Version 4.0 (December 2017)
- Version 3.0 (May 2015)
- Version 2.0 (January 2014)
- Version 1.0 (January 2011)

Version 5.0 Features (November 2018)

General

- Graphic wire-frame in picture view mode
- Project Manager: Categories remember last open/collapsed state
- Project Manager: drop-lists are activated by clicking anywhere, not just on arrow
- Improved warning message formatting
- Improved Print Preview display for graphics
- Column-line graphics are "filtered" to avoid clutter
- Improved area load graphics with shaded tops
- Graphics performance is **up to 20x faster**
- Removed 'memory leaks', which slowed program over time

Modeling

- Full area support (rigid and flexible) can now be applied to an area
- Ability to insert a vertex along an area side
- DXF import shows bounds and an option for centering at the origin
- Polygon boundary defined by side length in addition to radius

Loading

- Ability to move area loads by side coordinates
- Line stiffener loads can now be applied relative to wall orientation (parallel and perpendicular)
- Multiple selection in Load Case Manager

Analysis

- Line supports now insert a "drilling" spring support
- Individual service cases can be analyzed
- Analysis is now **12% faster**

Reporting

- **Name filters** are now enabled for both graphic and report filters
- Unavailable tables shown disabled with reason for not being available
- Table drop position used to locate when adding new tables in Text Reports
- Full report can now have categories filtered (model, loads, results, graphics)

Version 4.0 (December 2017)

The concrete design module of VisualPlate 3.0 was removed and ConcreteBending 4.0 was created

General

- 64-bit implementation, no more out-of-memory issues
- Multiple-threaded architecture uses all processor cores
- New UI, Ribbon toolbar, consistent with other IES tools
- 3D model/viewing (ctrl+mouse wheel or click the cube)
- Preference settings (fonts, colors, sizes, options, etc.)
- History Flies (automatic daily backups of a project-file, see preferences)
- Your Logo in a Text Report (see preferences)

Modeling

- No merge/intersection necessary for boundaries
- Easier to edit
- Automatic split/connect meshing for crossing or overlapping line supports, beams
- Control over overlapping for plate thicknesses
- DXF import has insertion point so huge coordinate files can be moved.

Loading

- Implements ASCE 7 load combinations
- More Direct full-boundary loading

Analysis

- Automated behind-the-scenes
- Much faster analysis (using all processor cores)
- Accurate results

Reporting

- Powerful Report Viewer
- Many tables and options
- Saved reports in project files
- Paste graphics into reports
- Complete project report
- Predefined, customizable reports

1.6 Preferences

VisualPlate preferences are default settings that primarily affect the behavior of new projects. These are not project-specific settings, which are found in the Project Manager. The preference settings can be adjusted through **Home | Preferences**. Some settings do not take effect until a new project is created or until the program is restarted. Use the Restore All Defaults button to restore the VisualPlate preference settings to their original state. While most of the preference settings are self-explanatory, a few are documented below. Preference settings are saved on your machine in the IES folder: *C:\Users\<your.login>\AppData\Local\IES\Customer*.

Project

The project preferences affect new projects, and do not affect the current project. For current projects, use the settings in **Project Manager**.

- **History Files** - Set how many once-per-day backup files VisualPlate should keep. Files are located in **Home | Custom Data** in the History Projects folder.
- **Next Inspector Field On Enter** - In the Modify tab, 'Enter' can simply accept changes or also advance like a 'Tab' to the next row.

Data

Set the default name prefixes for line stiffeners, line supports, and areas. These settings apply to the first objects created in a new project or immediately after restarting VisualPlate.

Fonts

Change the character size and styles used to display text in graphic views and reports.

Reports

- **Member Results Offsets** - Input the number of interpolated result locations along the member in some results tables (2-201).
- **Customer Logo** - Specify the location of a logo to use for the reports. If left blank,

AppData\Local\IES\Customer\ReportLogo.jpg will be tried.

- **Logo Alignment** - Select the alignment of the logo in the header.
- **Maximum Page Count** - Input the number of pages allowed in a report before a warning and truncation occurs.
- **Header Height** - Set the height of the header in the report.
- **Justification, Text Data** - Specify the justification of the text data in the report tables.
- **Justification, Physical Data** - Specify the justification of the physical data in the report tables.

Graphics

- **Graphic Sizes** - Change how large objects are drawn in graphic views.
- **Rotate, Pan, & Zoom -** Control how much or how fast the view changes with mouse-wheel or arrow keys.
- **Default Snap Points** - Set the default number of evenly spaced internal points that can be snapped to.
- **Print Resolution** - Set the DPI (dots per inch) precision to be used when displaying graphics views on the printer or when placing graphics information on the clipboard.

Colors

Change the colors of objects in Graphic Views and Reports. Every visible object type shown in graphic views have a default color (e.g. line stiffeners are blue by default).

1.7 Boundaries

Plate boundaries are the primary elements modeled and analyzed in VisualPlate. Boundaries with a wide variety of shapes and sizes can be modeled in the program. The boundary can be supported by Point Supports and Line Support and stiffness can be added by modeling Line Stiffeners into the boundary. A single material is used for all boundaries in the project and is defined in the **Project Settings**.

Modeling

In the Model and Load view, circular, rectangular, polygonal, or custom boundaries can be drawn using the buttons in the ribbon. Boundaries are defined by the vertices at boundary points. New boundaries can be drawn on grids or existing vertices and snap points can be used to create the boundary. The number of snap points along each edge of the boundary is specified in the in the **Project Manager | Filter** tab. Any arbitrary geometry can be constructed by drawing multiple boundaries connected to or overlapping each other. Adjoining boundaries are modeled as continuous, with common displacements and flexural rotations at the boundaries. Each individual boundary can have a different thickness. At locations of overlapping boundaries, the thickness is specified using the Thickness Overlap parameter in the **Project Manager | Modify** tab. Note: Disconnected boundaries are not allowed in VisualPlate and expansion-joints or other discontinuities cannot be modeled.

Holes

A selected boundary can be turned into a hole by setting Hole? = Yes in the **Project Manager | Modify** tab. Loads that exist within holes are not included in the analysis. If portions of Area Loads lie partially over holes, only the loading lying over boundaries are considered (i.e. the loading that occurs over holes is not distributed to adjacent plate elements)

Loading

Boundaries may be loaded with Full Area, Circular, Rectangular, Tubular, and Ring area loads or by concentrated Point loads. Loads can also be transferred to the slab by Line Stiffeners.

Analysis

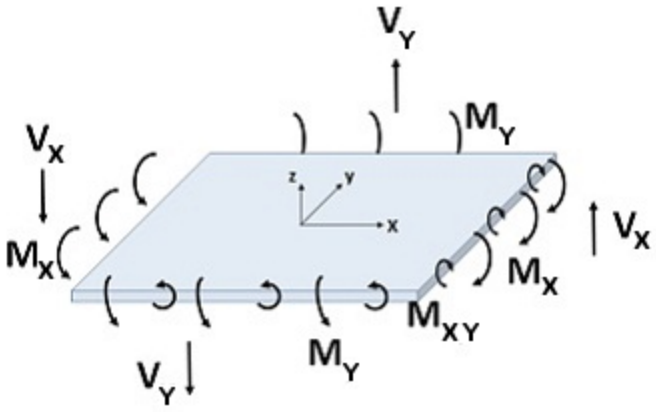
Finite element analysis is approximate and numerical. Use mesh refinement and examine results carefully, before trusting analysis results.

FEA Model

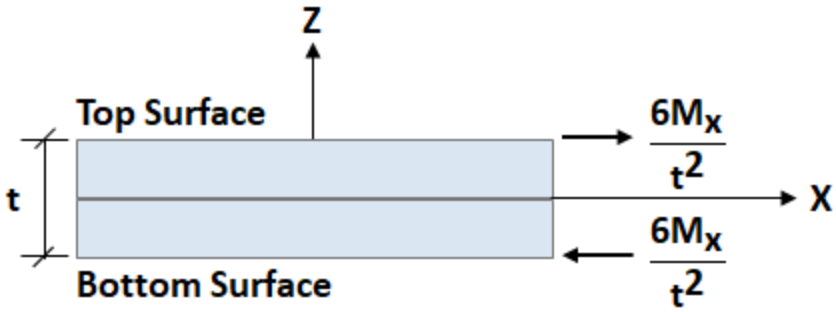
Boundaries in VisualPlate are meshed into triangular FEA plate elements. See the Analysis page for more information on the FEA mesh density, including the available settings and options. For a better understanding of the FEA model that is built by VisualPlate, use the **File | Export a VA Project** feature to create a VisualAnalysis project and examine the FEA model in more detail. More information on integration between VisualPlate and other IES tools can be found on the Integration page of the Help File.

Plate Results Sign Convention

1. Positive moments put the slab's top bars (Global +Z) in tension.
2. MX and VX moments and shears act on the Global X-face of the plate elements.
3. MY and VY moments and shears act on the Global Y-face of the plate elements.
4. Tensile stresses are positive and compressive stresses are negative.



Plates are only subject to bending stresses in VisualPlate. Therefore, the mid-plane is the neutral axis and the normal stresses at this plane are zero. The stresses at the top and bottom surfaces are equal in magnitude and opposite in sign.



Viewing Analysis Results

The results from the finite element analysis can be displayed graphically in the Analysis Results view. The **Project Manager | Results** tab displays the numerical results that correspond to the colored graphics. With nothing selected, the results displayed in the **Project Manager** are a summary for the selected result case, whereas if one or more individual plates are selected, the **Project Manager** shows the result range for the selected plates. The various Result

Cases that were included in the finite element analysis can be selected using the Result Case drop down from the **Ribbon | Home** tab. The Result Type displayed graphically in the Analysis Result view is specified in the **Project Manager | Result Filter tab**.

Plate Diagrams

Plate result moment, shear, and displacement diagrams are available in the Analysis Results view. Left-click and drag a line across the slab to view the result at the line's location or right-click and select Show Plate Result Diagram from the context menu to view the diagrams for the selected result case. The slice direction and location can be adjusted within the Plate Result Diagrams dialog box along with the number of plots displayed and the plot type. Note: The result location and plot settings will persist within one session of VisualPlate. This allows the Plate Result Diagrams dialog box to be closed to select a different result case or to modify the model without losing the adjustments that were made in the dialog.

Reporting Analysis Results

Analysis results can also be viewed in text form using the Text Reports tab. Once on the Text Report tab, a list of available tables is shown on the **Project Manager | Tables** tab and can be added to the text report by double-clicking an individual table or dragging and dropping a table into the report. Plates can also be reported on an individual basis by selecting one or more plates from the Analysis Results view, and using the right-click context menu to *Report Selected*.

1.8 Point Supports

Point Supports in VisualPlate are used to support the plate at a specified location in the model.

Modeling

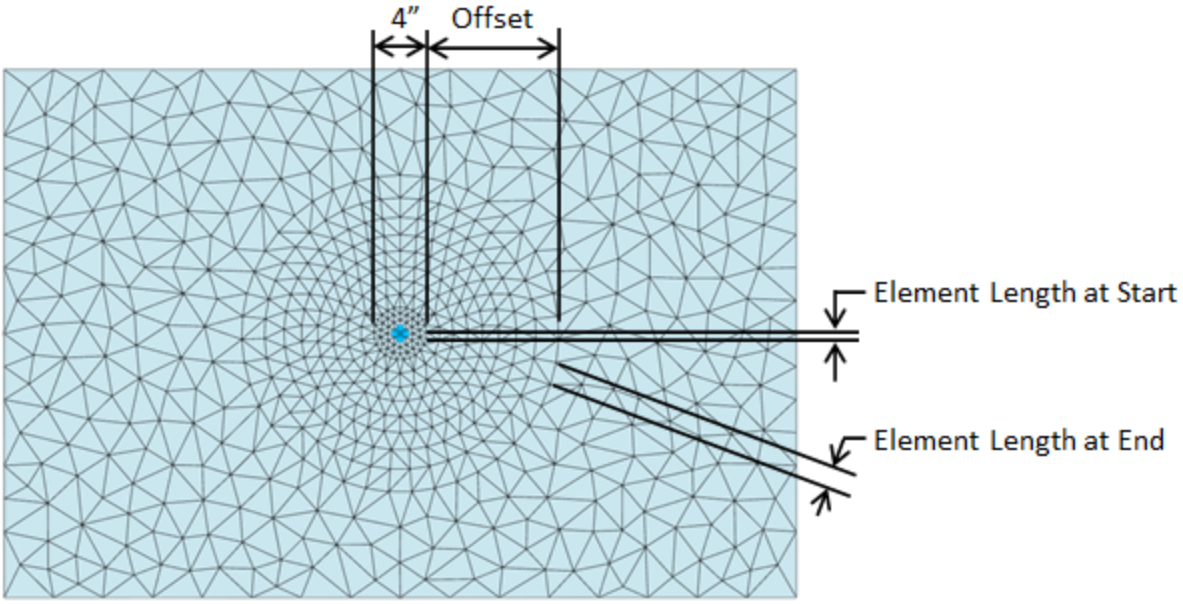
Point Supports are defined by their location in the XY-plane in the model. Note: In the Finite Element Analysis (FEA) model, only a single node in the slab's mesh is restrained at a point support; therefore, care should be taken when using a point supports to model a physical support that has a large cross-sections.

Support Type

VisualPlate only allows deflection in the Z-direction and rotation about the X-axis and Y-axis. Therefore, Point Supports have the option of being set to Fixed, Free, or Specified-K (stiffness) for Force-Z, Moment-X, and Moment-Y.

Mesh Refinement

The FEA method is approximate and the accuracy of the solution typically increases as the mesh becomes finer. Generally, a finer mesh should be used in regions where there are large changes in stresses in the plates (such as at point supports where the support can create stress concentrations). Select a point support(s) and set Refine? = True in the **Project Manager | Modify** tab to refine the mesh in the regions of the chosen point support(s). The mesh refinement parameters for point supports are defined in the image below. Note: Since point supports are a single point in the model and do not have physical sized, the offset and element length at start begin at a 4 inch diameter circle that encloses the point support.



Point Support Mesh Refinement

Loading

Point Supports cannot be directly loaded in VisualPlate. A Load Point can be defined at the same location as the point support (i.e. connected to the same node in the finite element model) and load can be applied to the Load Point.

Analysis

The force in the Z-direction and the moments in the X-direction and the Y-direction are calculated for each load case and reported for the Point Supports in VisualPlate. The displacement in the Z-direction and the rotations about the X-axis and the Y-axis are also reported.

Reports

Report tables, which include modeling and result information for the Point Supports, are available in the Text Report view.

1.9 Line Supports

Line Supports are used in VisualPlate to restrain the model along a series of nodes that fall on a straight line which is defined by a start coordinate and an end coordinate in the XY-plane.

Support Properties

Line Supports only provide restraint along a single line of nodes in the FEA model. VisualPlate only allows deflection in the Z-direction and rotation about the X-axis and Y-axis. Line Supports fix the deflection of the nodes on the line in the Z-direction and fix the rotation of the nodes about a line in the XY-plane perpendicular to the Line Support. The three support options (Pinned, Fixed, and Spring) allow the rotational restraint about the line defined by the Line Support in the XY-plane to be specified.

Analysis Results

The total force in the Z-direction and the total moment about the Line Support is calculated and reported for each load case.

1.10 Line Stiffeners

Modeling

Line Stiffeners are created using the **Boundary and Support | Draw Line Stiffeners** command to draw line stiffeners on top of an existing plate boundary in the model. Line stiffeners distribute applied loads to the plate boundary and add stiffness to the model. Material Properties are defined for the selected line stiffener(s) in the **Project Manager | Modify** tab. Line stiffeners that adjoin one another, regardless of orientation, are modeled such that load transfer (shear and moment) occurs between the members (i.e. there are no end release options for line stiffeners).

Shape Properties

The shape for a line stiffener member is defined on the **Project Manager | Modify** tab when one or more stiffener is selected. There are four types of shapes available for use in VisualPlate: Database Shapes, Standard Parametric Shapes, Custom Shapes, and Analysis Blobs. Custom Shapes are created in IES ShapeBuilder and added them to the Custom Shape Database.

Database Shapes

IES includes a large shape database of steel, wood, aluminum, cold-formed, and other shapes common in the USA and some other countries. The database is customizable using IES ShapeBuilder, but cannot be modified directly using VisualPlate. Selecting a database shape will typically also define material and therefore it is best to define the shape prior to defining the material. The shape database contains Virtual Joists and Virtual Joist Girders which are developed by the Steel Joist Institute. Their website has information on the basic concept and purpose. You may create models with these shapes as if they were steel beams (please understand their purpose and limitations before using them).

Standard Parametric Shapes

Parametric shapes are defined by dimensions such as width, depth, and thickness, which are input by the user. Parametric shapes are commonly used for defining concrete members (e.g. square, rectangle, round, etc.). VisualPlate offers the following types of parametric shapes: Angle, Channel, Circle, I-Shape, Pipe, Rectangle, Rectangular Tube, Spandrel, Tee, and Zee.

Custom "Blobs"

Custom Blobs are a quick way to defined the numerical properties of a shape that are needed to perform the analysis (the actual dimensions of the shape are not defined). Custom Blobs are only available in the project in which they were created (i.e. they are not added to the shape database). Use Custom Blobs with care as the shape properties are not checked other than to ensure they are positive.

Quick Shape Pick List

The **Project Manager | Modify** tab shows the types of shapes available in the Source drop-down list: Standard Parametric, Database Shape, <Add Custom 'Blob'...>, <ShapeBuilder>, followed by a list of shapes that are already used

or have been recently used in the project. Use this list to quickly find a shape for a new line stiffeners.

Loading

Vertical forces and moments in two directions can be applied to line stiffeners. The self weight of the line stiffener can be included by checking the option for the selected stiffener in the **Project Manager | Modify** tab. Line stiffener loads can be entered as distributed evenly along the member length or as the resultant total. Loads can be entered in the global coordinate system or in the line stiffener's local coordinate system (defined as parallel or perpendicular to the member).

Analysis

In the finite element model, line stiffeners are modeled using member elements in the same plane as the plate elements used to model the boundary. The stiffness of the line stiffeners are adjusted based on the Vertical Offset setting (i.e. the parallel axis theorem is used to modify the stiffness).

Displaying Results Graphically

The line stiffener results from the finite element analysis are displayed graphically in the Analysis Results view. The **Project Manager | Results** tab displays the numerical results that correspond to the colored graphics. The line stiffener's displacement, shear force, moment, and stresses can be viewed by changing the Result Type drop-down menu under the Line Stiffener Results category of the **Project Manager | Result Filter** tab. With nothing selected, the results displayed in the **Project Manager | Results** tab are a summary for the selected result case, whereas if a single line stiffener is selected, the Project Manager shows the result range for the selected line stiffener. The various Result Cases that were included in the finite element analysis can be selected using the *Result Case* drop down from the **Ribbon | Home** tab. Furthermore, moment, shear, and displacement diagrams for line stiffener(s) for each result case can be viewed using the Member Graph tab.

Reporting Line Stiffener Results

Analysis Results can be viewed in text form using the Text Reports tab. Once on the Text Report tab, a list of available tables is shown on the **Project Manager | Tables** tab and can be added to the text report by double-clicking an individual table or dragging and dropping a table into the report.

1.11 Load Combination Criteria

Load Combination Sets

Several sets of building code load combinations are built into VisualPlate. Custom load combinations may also be created in the Load Combinations tab of the Load Case Manager. Service level load combinations may be added if you want to look at results.

Importing Load Combinations

Custom factored load combinations can be imported from the clipboard using the **Import From Clipboard** button in the Load Combinations tab in the Load Case Manager. Text must be tab delimited and copied to the clipboard in the following format:

{ComboName} {Factor} {ServiceCaseName} {Factor} {ServiceCaseName2} ...

For example:

ComboName	1.2	D	1.6	L	0.5	Lr

MyCombo	0.9	D	1.3	W

Seismic Criteria

To correctly generate load combinations that contain seismic loads, several additional parameters are required. Please refer to ASCE 7, Section 12.4, for how these parameters are used in generating load combinations. It is important to note that these parameters (such as SDS and SD1) are only used to generate load combinations. For example, ASCE 7 says that Seismic Category A does not require the combined orthogonal direction combinations (e.g. X+30%Y), but Category D does. VisualPlate does not automatically generate any loads, just the combination cases.

1.12 Loads

In VisualPlate, loads are applied to the model in a service load case. The appropriate service case can be selected in the Service Case drop-down list in the Home or Loads ribbon. Loads may be easier to see and select by rotating the view to get a 3D perspective of the model. Note: VisualPlate only allows deflection in the Z-direction and rotation about the X-axis and Y-axis.

Concentrated Loads

Concentrated loads are applied to Load Points by selecting the Load Point(s) and clicking the Apply Load button in the Loads ribbon. The concentrated load consists of a force in the global Z-direction and moments in the global X-direction and Y-direction.

Line Loads

Line loads are applied to Line Stiffeners by selecting the Line Stiffener(s) and clicking the Apply Load button in the Loads ribbon. The line load consists of a force in the global Z-direction and/or moments which can be defined in the global X-direction and Y-direction or defined parallel and perpendicular to the line stiffener (i.e. in the line stiffener's local direction). Line loads can be entered as resultants or as distributed loads in the model.

Area Loads

Partial Boundary Loads

Rectangular, circular, tubular, and ring loads are used to apply pressure loads and/or overturning moments to some portion of the Boundary in the model. Pressures may be uniform or linearly varying in the global X-direction or global Y-direction. First set the Area Load type in the Loads ribbon and then sketch the load on the drawing grid in the model. The size and location of the load can be modified if needed after the load has been generated. Note: Loads will not be applied to holes in the plate's boundary.

Complete Boundary Loads

Select one or more Boundary and use the Apply Load command in the Loads ribbon to load the selected boundary. Alternatively, the Apply Multiple Boundary Load command in the Loads ribbon can be use to create a load that is applied to all boundaries in the model. Note: Boundary loads will automatically adjust to load the entire boundary if the boundaries's size or location is modified.

1.13 Load Points

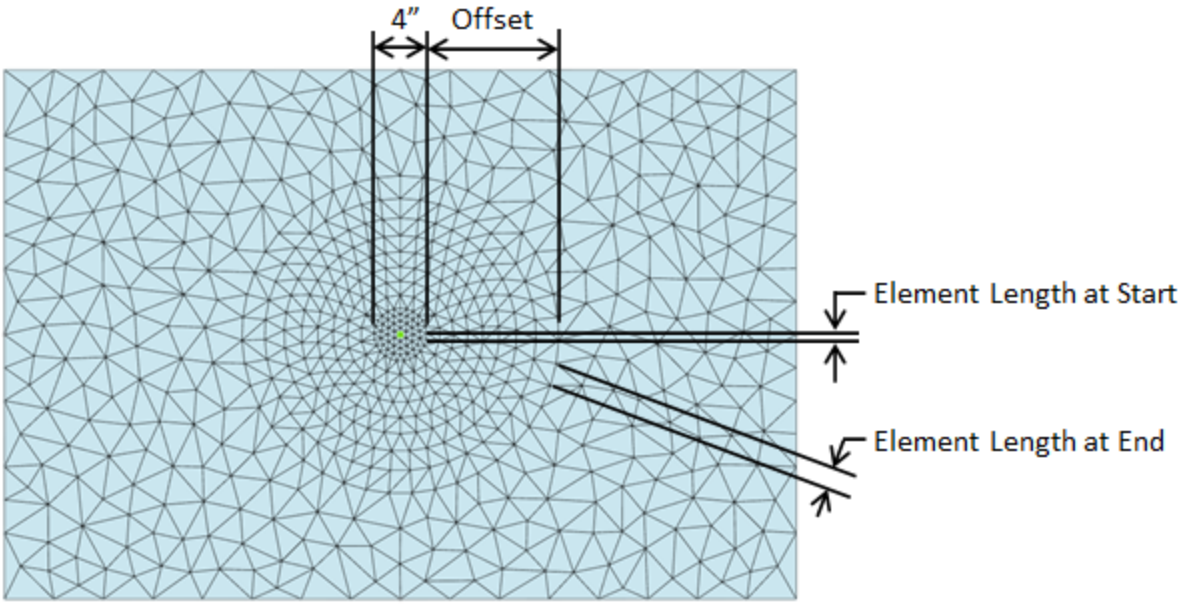
Load Points in VisualPlate are used to apply concentrated loads and moments to plates at a specified location.

Modeling

Load points are defined by their location in the model. Note: In the Finite Element Analysis (FEA) model, a single node in the slab's mesh is loaded; therefore, care should be taken when using a load point to model a physical load that has a large cross-sections.

Mesh Refinement

The FEA method is approximate and the accuracy of the solution typically increases as the mesh becomes finer. Generally, a finer mesh should be used in regions where there are large changes in stresses in the plates (such as at load points where concentrated loads can create stress concentrations). Select a load point(s) and set Refine? = True in the **Project Manager | Modify** tab to refine the mesh in the regions of the chosen load point(s). The mesh refinement parameters for load points are defined in the image below. Note: Since load points are a single point in the model and do not have physical sized, the offset and element length at start begin at a 4 inch diameter circle that encloses the load point.



Load Point Mesh Refinement

Loading

Concentrated forces (compression and tension), in addition to moments in two directions can be applied to load points.

Analysis

During the finite element analysis each load point is modeled as a single node connected to the plate's finite element mesh. Therefore, care should be taken when using a load point to model a physical load that has a large cross-sections.

Report

Multiple report tables, which include modeling, loading, and result information for the Load Points, are available in the Text Report view.

1.14 Material Properties

The material properties for plates and line stiffeners are defined using materials from either the IES Material Databases or the Custom Material Database as discussed below.

IES Material Databases

Several common material databases are included with VisualPlate listed under IES in the Material Database dialog box. These materials can not be modified or removed from the system. Contact IES Technical Support to suggest additional libraries of materials for IES to include in the database.

Custom Material Databases

Custom materials can be created in VisualPlate which are stored in the Custom Data Files. To add a custom material to the database in VisualPlate, click the Add Custom Material button in the Material Database dialog box. Use a General Material Type or choose the appropriate Material Type and edit the Defining Properties as needed. Custom materials can be used by VisualPlate, VisualAnalysis, ShapeBuilder, and other IES products (except Quick-products).

Limitations

All IES materials are assumed to be linear, isotropic and elastic. VisualPlate makes common use of orthotropic materials, like wood, but does so using isotropic properties. VisualPlate utilizes four primary properties: modulus of elasticity, Poisson's ratio, thermal coefficient of expansion, and weight density. Specific material types have additional properties that need to be defined (e.g. steel has yield stress, concrete has compressive strength, etc.).

Data Format

The database consists of XML data files. XML is a text based format which is commonly used for data-exchange and can be edited easily using a simple text editor (NotePad++ is recommended since it is freeware and offers XML syntax highlighting). Microsoft Word and similar programs are not recommend as they have a tenancy to corrupt the format.

Material Management

Look for instructions and examples, in the Customer\Materials folder. Database files can be copied to other computers where IES products are installed. Files that are no longer needed can be deleted. Files can be edited to change data or to remove shapes or shape categories. Make sure to backup the customized files.

1.15 Self Weight

In VisualPlate, the Z-axis is defined as the vertical axis. Therefore, gravity can only act perpendicular to the plate boundaries defined in the model. The self-weight of the elements can be set on an individual basis.

Plates and Line Stiffeners

To include a plate or line stiffener's weight in the Dead-Load service case, switch to the Model and Load view, select the object, and check the Add Self Weight box in the Project Manager | Modify tab. The item's density is determined by the selected material.

Point Supports, Line Supports, and Load Points

These items do not have self-weight.

1.16 Analysis

VisualPlate uses finite element analysis (FEA) to determine the displacements, shears, and moments in the plate system. Exporting the model as a VisualAnalysis project (**File | Export a VA Project**) will show what is happening behind the scenes in VisualPlate. While VisualPlate attempts to handle many details of the FEA method, some knowledge of FEA is still required to use the program successfully.

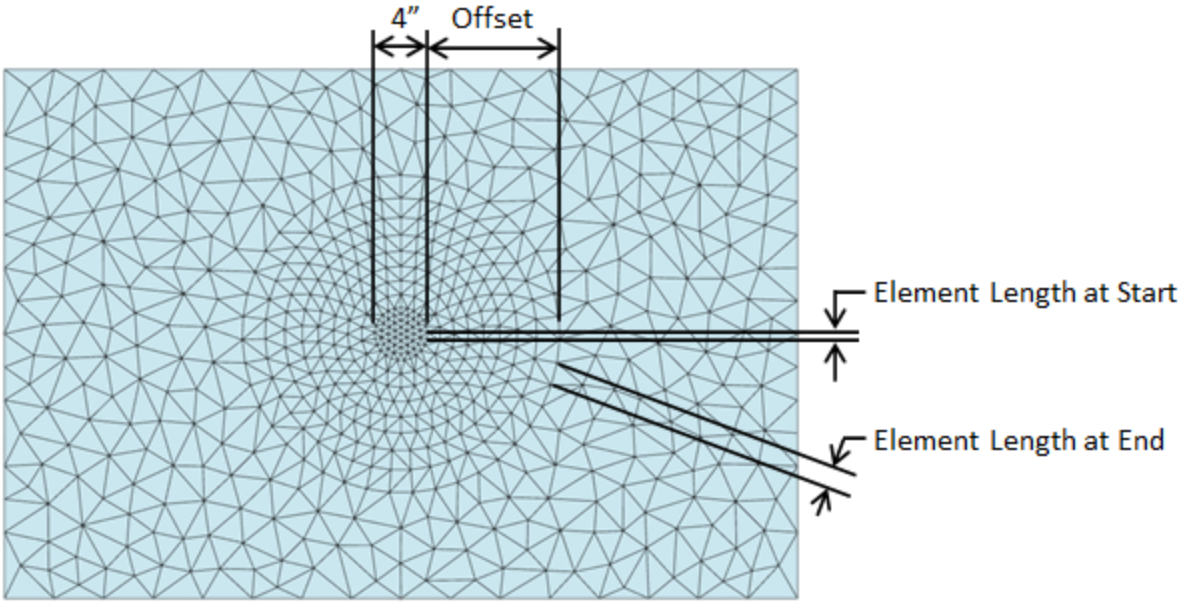
Mesh Settings

To control the number of plate elements, pick between course, medium, fine, or specify a desired number directly in the **Project Settings**. The actual number of meshed plates generated in the model is shown under Meshed Plates. Turn on Meshed Plates in the **Project Manager | Filter** tab to view the meshed plates in the model. In VisualPlate, the advanced finite element meshing options can be modified if needed. Note: The mesh settings at Load Points and Point Supports can by set by selecting the model object(s) and adjusting Mesh Refinement settings in the **Project Manager | Modify** tab.

Mesh Refinement

It is the user's responsibility to verify and validate the results obtained from VisualPlate. The FEA method is approximate, and the accuracy of the solution depends on how fine the mesh is in the model (generally, a finer mesh produces more accurate results). A finite element mesh refers to the multiple plates that are used to model a single component (such as a plate boundary). Mesh Refinement is the process of reanalyzing the model with successively finer and finer meshes and comparing the results between these different meshes. As the mesh is refined, the change in the solution becomes smaller and an asymptotic behavior of the solution starts to emerge as shown in the figure under Mesh Refinement Procedure below. Eventually, the changes to the solution will be small enough that engineering judgment can be used to determine that the model has converged.

In VisualPlate, mesh refinement is accomplished by reducing the Element Count in the **Project Manager | Modify | Project Settings**. Furthermore, as shown in the image below, the mesh can be refined at Load Points and Point Supports by selecting the model object(s) and adjusting Mesh Refinement settings in the **Project Manager | Modify** tab. In general, a finer mesh should be used in areas where there are large changes in stresses in the plates (e.g. load point locations that receive concentrated loads and point support locations where the support at a single node can create stress concentrations).



Load Point & Point Support Mesh Refinement

Mesh Refinement Procedure

1. Model the boundary and VisualPlate will automatically generate the plate elements based on the Mesh Refinement settings.
2. Let the analysis run and record the results.
3. Increase the number of Meshed Plates for the boundary and/or refine the mesh at Load Points and Point Supports as discussed above.
4. Let the analysis run and record the results.
5. Compare the results from Step 4 with the results from Step 2. If the difference in the analysis results is small and acceptable (using engineering judgment), the mesh refinement process is complete. If the difference in the analysis results is large and unacceptable (using engineering judgment), start back at Step 3.

Note: Model results such as displacement, moment, shear, etc., which are represented as Phi in the graph below, will coverage at different rates. Therefore, it is important to ensure that the model has converged for the result of interests.

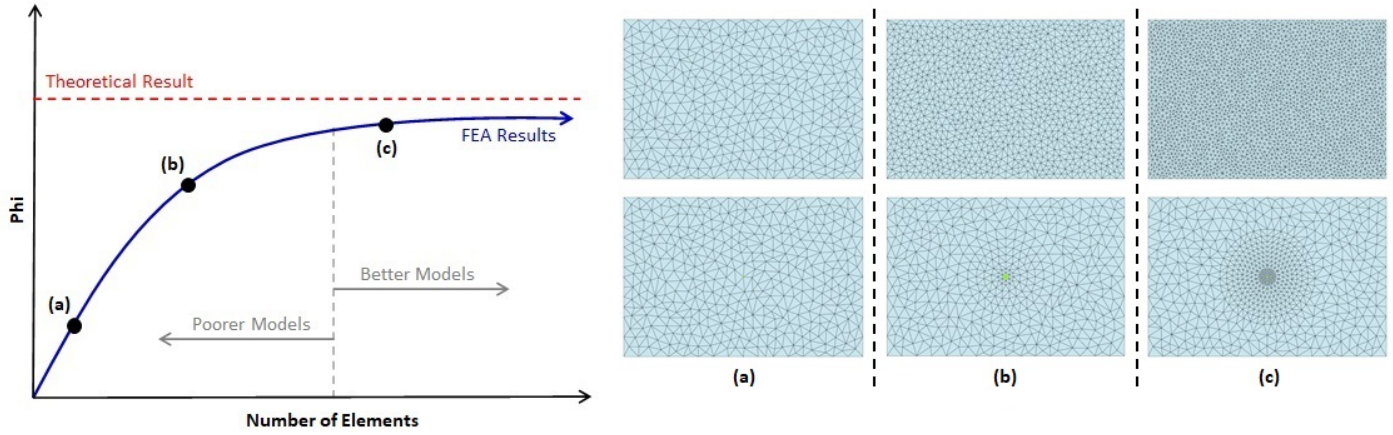


Plate Bending

The bending part of the FEA plate element is based on the triangle formulation originally presented by Xu et. al.[1] in 1992. This element accounts for transverse shear effects present in structures that might contain areas with thick plates, such as footings or thick floor slabs.

Statics Checks

A Statics Check is performed for each load case analyzed in VisualPlate. The total applied loads in each global direction is calculated and compared to the sum of all support reactions in the corresponding global directions. The applied loads are based on the deformed shape of the structure while the reactions are based on the structure's undeformed shape. If the loads and reaction are equal and opposite in magnitude, then the structure is in equilibrium. An imbalance indicates that the deflections are large enough to generate inaccurate results which might indicate that there is a modeling problem. VisualPlate provides a warning if a significant imbalance is detected. If a warning is received, carefully review the model to ensure it is set up correctly and verify the results. The Statics Check is displayed on the **Project Manager | Results** tab or the Statics Check Information table can be added to the report.

References

1. Xu, Zhongnian, "A thick-thin triangular plate element" *International Journal for Numerical Methods in Engineering*, Vol. 33, 1992, pp. 963-973.

1.17 Integration with IES Tools

VisualAnalysis

VisualPlate can be used to create a VisualAnalysis project file (.vap) using the **File | Export a VA Project** feature. The .vap file can be used to look at the finite element model that was created behind the scenes in VisualPlate or to perform more advanced analyses (such as a dynamic analysis) which are not supported in VisualPlate. Note: VisualAnalysis is licensed separately by IES. Contact Sales for additional information regarding this product.

1.18 Reports

Reports in VisualPlate are designed to present information in a clear, concise, and organized fashion. Reports can include both text-based and graphical information that can be printed to paper, to .pdf, or saved in a number of different file formats. Graphical information can be inserted into a report using the **Copy** and **Paste** commands or printed directly using the **File | Print** command.

Report Essentials

- Tables
- Member Graphs

Custom Report Logo

The report may be customized to include your own (company) logo in the header. All you need to do is create a logo image: ReportLogo.png or ReportLogo.jpg, and place it in the IES\Customer folder, which you can access via the **Tools | Custom Data** toolbar command. The image should be kept to less than 5 times wider than it is tall. It will be scaled to fit in the header area, but wide images may cause other text to start wrapping or get truncated. If the image works you'll see it in the report/preview immediately after restarting VisualPlate. This feature is also available in other IES tools.

1.19 Tables

In VisualPlate, tables are used to report information in a clear and concise manner. The tables available for the report are listed in the **Project Manager | Tables** tab when the Report View is active. Tables fall into one of five categories (Project, Structure, Load, and Result) and will automatically appear or disappear depending on the items in the model (elements, loads, etc.) and the available analysis results. Hover the mouse over a table in the list to view its description.

Table Types

- **Project Tables** are used to document the project wide information for the model including the Project Settings, Service Load Cases, and Factored Load Combinations.
- **Structure Tables** are used to document the input data for various model objects including Boundaries, Load Points, Point Supports, Line Supports, Line Stiffeners, etc. Also, a Model Summary can be reported.
- **Load Tables** are used to document every load applied to the model in each service load cases including Area Loads, Point Loads, Line Loads, etc.
- **Result Tables** are used to document the analysis results for the elements in the model including Plate Forces and Displacements, Beam Forces and Displacements, Point and Line Support Results, etc.

Adding & Removing Tables

To add a table to the report, simply ***drag*** the table from the **Project Manager | Tables** tab to the desired location in the report or ***double-click*** on the table to insert it at the end of the report. A list of the report's Included Tables is shown in

the **Project Manager | Tables** tab which can be rearranged by ***dragging*** them with the mouse. To remove a table from the report, click the X next to the table in the Included Tables list or ***right-click*** on the table in the report and select Remove.

Modifying Tables

Tables can be modified using the Report Settings or Model Filters in the **Project Manager | Report Filter** tab. The Report Settings are used to specify which Service Cases and Result Cases to include in the report while the Model Filters are used to filter the items that are included in the report (such as Boundaries, Point Supports, Load Points, Line Stiffeners, etc.). Tables can also be modified by clicking on the tables in the report. ***Click*** the column header to sort the column, ***drag*** the column header to rearrange the columns in the tables, or ***drag*** the column boarders to adjust the column widths.

Selected Table

Click within a table to select the table and activate the Selected Table section in the **Project Manager | Report Filter** tab. In this tab, the Title can be modified, the columns can be sorted, and the page width can be defined. Choose which columns are included in the table under the Columns section and ***drag*** the columns in this section to rearrange them in the report.

Selected Table Extremes

Certain tables have the Selected Table Extremes option available in the **Project Manager | Report Filter** tab. The following parameters are used to set how the information is filtered in the selected table.

- **Extreme Rows -** Set to show the Extreme Rows Only for the table or to Show All (which can lead to lengthy reports that may need to be filtered by result cases or reported items to be manageable).
- **Included Rows -** Specify how the extreme rows are considered.
    - **Max and Min -** Keep only the max and min values.
    - **Max -** Keep only the max value.
    - **Min -** Keep only the min value.
    - **Max/Min (when opposite sign)** - Keep the max and min values, if different signs, else keep the most extreme.
    - **Extreme -** Keep only the most extreme value, positive or negative.
- **Applies To -** Specify if the extreme rows be kept on a table wide basis or by each item in the table.
- **Consider Zero as Extreme -** Specify if zero should be considered an extreme value.
- **Show All Extreme Rows** - Choose to show all rows with the extreme value or only show the first occurrence of the extreme value.

Column Justification

Right click on a column in the report to set the Column Justification to Left, Center, or Right for an individual column in a table. In the Reports category of the Preferences, the default Justification for the Text data and Physical data can be specified.

1.20 Member Graphs

Member Graphs display detailed diagrams (displacement, moment, shear, and torsion) for line stiffeners along their length. The results can be displayed for one single line stiffener or for a chain of line stiffeners for a single Result Case.

Create a Single or Multi Member Graph

To create a single member graph, simply select a line stiffener in the Model and Load or Analysis Results view and switch to the Member Graph view. A different line stiffener can be chosen from the dropdown in the **Project Manager | Graph Filter** tab. To create a multi member graph, simply select two or more connected line stiffeners that form a line and switch to the Member Graph view. If the selected line stiffeners do not have local axes in the same direction, a note will appear on the graph indicating that the member chain has inconsistent local axes.

Customize a Member Graph

Member graphs can be customized in the **Project Manager | Graph Filter** tab. Annotations, Data Points, Grid Lines, and Shadows can be adjusted for the plots. Use the Details Dialog to change specific graphic format information like colors and fonts as well as the basic type of plot used.

Print a Member Graph

Member graphs can be printed directly using the **File | Print** command. The orientation (portrait or landscape) of the graphs can be set using the **File | Page Setup** dialog. Use **File | Print Preview** to view the page before printing.

Export a Member Graph

To export a Member Graph to another program, use the **Home | Copy** command to copy the Member Graph to the clipboard and then use **Paste** to insert the picture into the other application.

1.21 Support Resources

Did you Search this Help File?

Take advantage of the help and support built into the software, as described in the Program Layout section of the User's Guide. This document can be searched, and you should try different potential terms, sometimes less is more when searching (use just the unique word or words). A Table of Contents is also available.

Do Not Contact Support For:

- **Licensing/Sales.** Use www.iesweb.com or sales@iesweb.com.
- **Modeling Advice.** Determining how to model a structure is your responsibility as an engineer.
- **Model Validation.** IES cannot validate your model or your results. If you can document a software defect, contact support and we will investigate further and create fixes as necessary.
- **Engineering Theory**. IES is not in the business of educating engineers. There are textbooks referenced in this help file.

Technical Support

- **Support Email:** support@iesweb.com. Replies are usually within 2 business hours, if you don't hear anything within a business day, assume it got spam filtered or lost and follow-up. For best results, be sure to ask a question, indicate exactly which IES product & version you are using, include as much detail as is practical. If relevant, please attach a project file and/or screenshots.
- **Support Telephone:** Not Available. We have found this to be too inefficient for everybody. With email you can attach a screen shot, a project file, and we can better direct your question to the IES expert for that product or area. Phone tag takes longer than you think.
- **Business Questions:** For any licensing or sales-related questions or issues contact sales@iesweb.com.