ShapeBuilder 14.0

User's Guide

Updated: 3/27/2025

ShapeBuilder 14 User's Guide

Table of Contents

1 Introduction

1.1 Welcome to ShapeBuilder 14

ShapeBuilder will help you quickly and accurately determine the geometric and structural section properties of complex built-up or cut-down cross-sections. ShapeBuilder also performs a finite element analysis to determine the torsion properties and the shear stresses for any arbitrary cross-section. In addition to calculating a variety of section properties (area, moment of inertia, radius of gyration, section modulus, torsion constant, etc.), loads can be applied to the cross-section to determine the normal stresses and shear stresses. Shapes created in ShapeBuilder can be exported to the IES Shape Database for use in other IES applications, such as VisualAnalysis.

Getting Started

- Use **File | Open** Example 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

ShapeBuilder 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 ShapeBuilder on your computer, you become a registered user of the software. The ShapeBuilder 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 ShapeBuilder 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

Creating Shapes

- Create built-up sections to find their properties
- Use standard parametric shapes as building blocks
- Use shapes from a large database of steel, wood, cold-formed, aluminum, etc.
- Import polygon shapes from simple text files (.txt)
- Import shapes from CAD DXF/DWG files
- Import shapes from STEP/IGES files
- Mark any part as a hole
- Create shapes with multiple holes or cut-outs
- Create new shapes by copying existing items
- Define the material setting for each part
- Composite sections can be analyzed (i.e. transformed properties are calculated)
- Export Shapes to the Custom Shape Databases for use in ShapeBuilder and other IES products like VisualAnalysis

Analysis

- Numeric integration
- Finite element analysis
- Finite Element model is automatically constructed
- Finite element mesh is easily refined
- Analysis is performed in the background while you work

Simple Properties Calculated

- Geometric (area, moment of inertias, elastic section modulus, etc.)
- Principal (orientation of principal axes, moment of inertias, elastic section modulus, etc.)
- Polar (moment of inertia and radius of gyration)
- Plastic (location of plastic neutral axes and plastic section modulus)

Advanced Properties Calculated

- Normal Stresses (combined flexural and axial)
- Shear Flow and First Moment of Area
- Torsion Properties (shear center, warping constant, torsional moment of inertia, etc.)
- Shear Stresses (St. Venant, flexural, combined, etc.)

Stress Levels Calculated

- Axial force, bending moments, shear forces, and torques can be applied to the shape
- Normal stresses
- St. Venant shear stresses
- Flexural shear stresses
- Combined St. Venant and Flexural shear stresses

- Resultant shear stresses

Reporting

- Quick Full Report includes the results and all the graphics available (can be modified)
- Quick Basic Report includes the results and active graphic view (can be modified)
- 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

- 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 & Assumptions

General

- Uses constructed geometries to perform a numerical approximation for all section properties.
- Slight differences between ShapeBuilder and database values is expected due to tolerances, manufacturer minimum properties, and unpublished geometric details.
- 2D (plane) figure analysis. Length is not considered for the beam or column member.
- Does not perform member design or check design specifications
- Will not analyze cracked concrete sections (please see IES ConcreteSection for this ability)
- Does not produce structural drawings

Advanced Analysis

- Shear stresses and torsion properties are only available for shapes with a single outside boundary and a single material. Normal stresses are available for disconnected shapes.
- Multiple-part touching parts are assumed to be fully connected.
- Calculated stresses are based on elastic material properties (i.e. yielding is not considered).
- Does not calculate Torsional Stresses due to Warping. This requires more information about the full length of the member and the boundary conditions on the member, and the variation of loading along the length of the member. None of which is available in ShapeBuilder. You can use the Warping Normal Function, which is calculated, to help you determine warping stresses.
- Does not calculate stresses due to concentrated loads applied at a specific point or area of the cross section. The applied loads are treated as "body forces", even though they are referenced from a specific point.

ShapeBuilder looks at the general stress distribution in the member due to external loads applied, not at localized stresses produced from a point load. The purpose of the specific load application point is to account for eccentricities that would create secondary forces (e.g., an eccentric shear force produces an additional torsion on the cross section).

- Cannot perform a stress analysis from internal pressures or thermal variations in a cross section.
- Orthotropic material behavior is not supported. Isotropic material behavior is assumed and used (the same material properties are used in all directions).
- Visco-Elastic material analysis (creep) is not supported.
- Dynamic loads are not supported.

Reinforced Concrete

- Older versions of ShapeBuilder offered cracked section analysis of reinforced concrete shapes. These features were removed in ShapeBuilder 10.0 and put in ConcreteSection.

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 Support Resources.

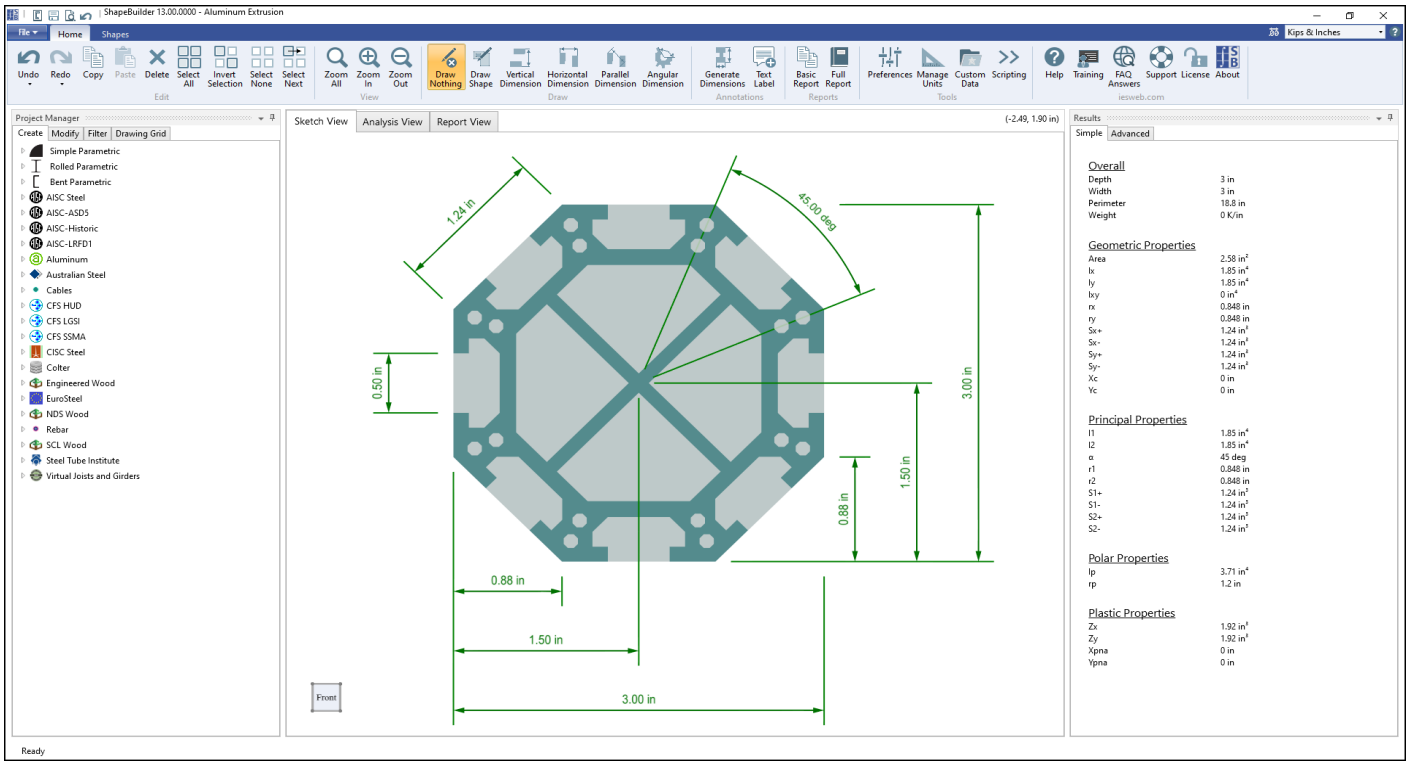
1.3 Program Layout

Explore

The best way to learn ShapeBuilder 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 the help or 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 your mouse over the screen image below for information about each area of the program.



Title Bar

The title bar displays the version of ShapeBuilder and the file name of the project. Also, there are helpful buttons in addition to the Windows system buttons.

Main Menu / Toolbar

Each command on the menu or toolbar is accessed with a click. They have helpful descriptions or tool-tips available by hovering the mouse over the command (this will also display the shortcut key command).

Project Manager

- The **Create** tab is used to add shapes to the model by double-clicking on shapes or dragging them onto sketch view.
- The **Modify** tab is used tab to change properties of selected shapes or to modify the Project Settings when nothing is selected.
- The **Filter** tabs is used to control what is shown or hidden in the active view.
- The **Drawing Grid** tab is used to define and show or hide drawing grids that are used to place or sketch shapes in the Sketch view.
- The **Analysis Settings** tab is used to define the loads, adjust the finite element mesh, and select the visible results.
- The **Tables** tab is used to add tables to the Report view
- The **Report Filter** tab is used to modify the tables in the Report view.

Graphic Views

These views provide a way to view the shape model, analysis results, and reports. Each tab displays different options and will provide different information in the Project Manager. The right-hand portion of the window-tab area contains

mouse coordinates. Text Labels and Dimensions can be added to the shape in the graphic view.

Results

This panel provides both the “simple” and the “advanced” results for the analyzed shape in separate tabs. These results update automatically as the shapes or analysis controls are modified. Hover the mouse over the name of a property for a pop-up tip that includes a description for that property.

Status Bar

Shows background meshing/analysis progress. Background processing is done on a separate thread of your processor, so you may continue working while they run. 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.

Unit Styles, Precision

In ShapeBuilder, you can display and enter physical quantities in a variety of unit-systems, including your own custom setup. Go to **Home | Manage Units** to select the units for the project from a list of common predefined unit styles or to create a custom style. The unit system can be changed using the drop-down box in the upper right corner. The button in the upper right corner allows you to change the number of decimals displayed after the decimal point, cycling between 0 and 7 digits.

Data Entry: Physical Quantities

When entering physical quantities, you may leave off the units to use the previously displayed units or enter different units (e.g. feet instead of inches). Also, mathematical expressions can be used when entering values (e.g. 5+2 in) and length units may be entered in "ft-in" notation. Entered values are converted and then redisplayed in the current unit style.

Mouse and Keyboard Commands

Hovering:

- Hover over a tool bar command for an explanation and keyboard shortcut
- Hover over a the name of an edit item in the Modify tab for an explanation
- Hover over a property names in the Results for a description of our terminology
- Hover over a part in the Sketch View to see which part will be selected if ***Clicked***
- Hover over a the shape's color-plot for stress values under the mouse point in Analysis View ("fly-by information" must be turned on in the filter)
- Hover anywhere in the Sketch View for coordinate locations (shown upper-right corner)

Selection:

- ***Click*** (on a part) to select the highlighted part under the mouse and unselects everything else
- ***Click*** in the 'whitespace' view to unselect everything
- ***Ctrl+Click*** to toggle part selection without affecting other parts
- ***Shift+Click*** to select all items of a given type
- ***Shift+Drag*** (away from parts) to draw a selection box (left-to right selects fully enclosed shapes, right-to-left selects any partially enclosed shapes)
- ***Shift+Ctrl+Click*** to select all shape parts with the same name prefix
- ***Shift+Ctrl+Drag*** (away from parts) to draw a selection box and select parts without affecting other parts

Move Part(s)

- ***Drag*** (press and hold the left button) to move a part, dimension, or label
- ***Drag*** on a part's vertex move the part by the vertex and snap it to another part's vertex
- ***Arrow Keys*** will move selected parts (the distance can be adjusted in the **Project Manager | FIlter** tab)
- ***Shift+Arrow Keys*** will move selected parts 4x the amount defined in the filter

Zoom:

- ***Scroll Mouse Wheel*** with the pointer over the point to zoom in or out from.
- ***Double-Click Mouse Wheel*** to zoom all/extents
- ***Ctrl+*** (plus) and ***Ctrl-*** (minus) keys.
- ***Ctrl+Home*** for zoom all/extents

Pan:

- ***Drag Mouse Wheel*** to pan.
- ***Arrow Keys*** will pan when nothing is selected
- ***Shift+Arrow*** keys will pan 2x the amount

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.

Hot Keys:

- ***Alt*** will expose the hot-keys in the main menu
- **F1** Help.
- **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.

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 14.0

Major Features

- Added rotate to horizontal/vertical ribbon and script commands to rotate shapes to align two selected points with the X or Y axis
- IX and IY for individual shapes about their centroid in the global coordinate system are now displayed in results and reports
- Added a normal stress by material report table to show the maximum and minimum normal stresses for each material
- Introduced the ExportResults() command for generating a tab-delimited text file of the calculated property results
- Added the GetPartProperty() command to return properties for specified parts
- Enhanced the GetSectionProperty() command to return values for E and Fy which are often needed of custom calculations
- Added the Buckling Equations example script to show how custom calculations can be performed in the program
- Improved dimension handling for easier repositioning selection when dimensions overlap with shapes

Minor Features

- Fixed various minor bugs in the program
- Updated Microsoft .NET framework for features and performance

1.5 Release History

Overview

- Version 14.0 (November 2023)
- Version 13.0 (August 2022)
- Version 12.0 (June 2021)
- Version 11.0 (January 2019)
- Version 10.0 (February 2018)
- Version 9.0 (September 2016)
- Version 8.0 (January 2015)
- Version 7.0 (October 2013)
- Version 6.0 (May 2011)
- Version 5.0 (January 2010)
- Version 4.5 (July 2008)
- Version 4.0 (December 2004)
- Version 3.0 (June 2002)
- Version 2.0 (July 1998)
- Version 1.0 (January 1996)

- Version 14.0 (November 2023)
- Version 13.0 (August 2022)
- Version 12.0 (June 2021)
- Version 11.0 (January 2019)
- Version 10.0 (February 2018)

- Version 9.0 (September 2016)
- Version 8.0 (January 2015)
- Version 7.0 (October 2013)
- Version 6.0 (May 2011)
- Version 5.0 (January 2010)
- Version 4.5 (July 2008)
- Version 4.0 (December 2004)
- Version 3.0 (June 2002)
- Version 2.0 (July 1998)
- Version 1.0 (January 1996)

Version 14

Major Features

- Added rotate to horizontal/vertical ribbon and script commands to rotate shapes to align two selected points with the X or Y axis
- IX and IY for individual shapes about their centroid in the global coordinate system are now displayed in results and reports
- Added a normal stress by material report table to show the maximum and minimum normal stresses for each material
- Introduced the ExportResults() command for generating a tab-delimited text file of the calculated property results
- Enhanced the GetSectionProperty() command to return values for E and Fy which are often needed of custom calculations
- Added the Buckling Equations example script to show how custom calculations can be performed in the program
- Improved dimension handling for easier repositioning selection when dimensions overlap with shapes

Minor Features

- Fixed various minor bugs in the program
- Updated Microsoft .NET framework for features and performance

Version 13

Scripts & Command Line

- Command Line is a powerful new way to drive the program:
  - Import CAD files with options to scale shapes, specify materials, etc.
  - Add database shapes, parametric, or custom shapes
  - Move and rotate shapes, zero the centroid, set the hole parameter, etc.
  - Apply axial load, shears, moments, and torsion to the cross-section in various coordinate systems

        - Refine the finite element mesh
        - Extract section properties, normal stresses, shear stresses, etc.
        - Add tables to report, export reports, write section properties to tables
    - External Script files can automate common tasks and much more:
        - Refine the mesh until the desired section property converges
        - Optimize shape sizes to carry specified loads
        - Batch process CAD files, generating ShapeBuilder files, reports, and a section properties table

Import CAD

- DXF/DWG Import Options dialog box overhaul
- CAD Import now supports Polylines, 2D Polylines, 3D Polylines, Circles, Ellipses, 3D Faces, and Solid objects
- Polyline are no longer required to have their "Closed" property set to Yes in the CAD file to import into ShapeBuilder
- The Import CAD default material now keys off the User Preference setting

Version 12

Major Features

- Shape Database Overhaul
    - Add/Edit/Delete functions are now available for custom databases, categories, and shapes
    - Custom database shapes can be imported from or exported to the clipboard
    - Improved database loading performance
    - Shape database re-write improves over all program performance

Minor Features

- Bent zee with lips added to parametric shapes
- Bent eave strut added to parametric shapes
- Side Points can now be modified in the inspector
- Default material preference added for simple parametric shapes
- Aluminum shape database ADM 2020 update
- Added Steel Tube Institute HSS Design Manual Volume One shape database
- Set default materials for timber shapes
- Import CFS library now prompts before overwriting an existing file
- Drawing grid improvements
- Meshing improvements at shared shape boundaries
- Improved report footers with cleaner table breaks across pages
- Improved responsiveness of the user interface
- Crash reporting and recovery improvements

Documentation

- Database Customization Help File and Training Video Documentation

- ShapeBuilder 12.0 Upgrade Guide Training Video

Version 11

General

- Shape boundaries can now be sketched
- Drawing grids are now available in the Sketch View
- Graphics are new for the entire program and are clearer, faster, and more reliable
- Project Manager: Categories remember last open/collapsed state
- Project Manager: drop-lists are activated by clicking anywhere, not just on arrow
- Improved Print Preview display for graphics
- Removed 'memory leaks', which slowed program over time
- Start screen shows thumbnail views of recent projects
- The filter state is now saved with the project

Modeling

- Flattened model is no longer shown in the Sketch View and is now exclusively shown in the Analysis View
- DXF import shows bounds and an option for centering at the origin
- Angular dimensions have been improved

Analysis

- Both the Shear Flow and First Moment of Area are calculated at user-specified locations (horizontal or vertical line) and for each part in the model
- Plastic properties available for more material types
- Result radio buttons have been added that automatically activate making it apparent which results are available

Documentation

- The Help File has been updated and is now available online and in a PDF format
- New training videos have been created for ShapeBuilder 11.0 that cover more topics

Version 10

Engineering Benefits

- Optional Manual Shape Flattening: for problems like parts in holes, or nested shapes
- Bending + Axial Stresses for composite sections (Flitch beam)
- **Report View** added for fully customizable reporting
- Import and export Auto-CAD **DWG** files directly
- **Import flexibility** with CAD files (layers and other options)
- **Part properties** are available for all shapes, and also in reports
- Advanced FEA mesh settings to help with unusual geometry

- Graphics now allows **'unblended' stress results** (for debugging FEA issues)
- Smarter use of tolerances allows very small shapes to analyze

Usability Benefits

- Easy result **export to text file**
- Improved **analysis performance** (5x faster in some cases)
- **Start screen** is easier to use, advertises features and training videos
- **Subtract** overlapping holes from a part
- **Move** (Up, Down, Left, Right) buttons added to Shape menu
- **Rotate about Point** command now can use a pre-selected vertex
- **History File** mechanism: automatically backs up project files in Custom Data folder
- Automatic **crash-backup file** mechanism
- Shift+Ctrl Click uses name-prefix selection on shape parts
- **Simplified menu**, requires less tab-switching
- Updated to latest compilers & tools
- Crash-prevention work
- Better validation and testing
- Updated documentation

Removed Features

- **Cracked concrete analysis** and interaction diagrams were moved to: ConcreteSection. If you had a license for ShapeBuilder 9.0, you may be entitled to a free license of ConcreteSection.
- **Legacy support**: This version no longer works with the obsolete databases from VA 12.0, SB 6.0 and prior versions. Nor will it open SB 6.0 or prior project files. You may use ShapeBuilder 9.0 in the rare case you need this ability. Saving a project file in that version will upgrade it to a usable format.

Version 9

Section Properties

- Unit Weight is now reported, helpful for complex or composite shapes.

Functions/Operations

- **Parts inside of holes** are handled much more intelligently!
- **Split a Part** Horizontally or Vertically at the location you choose
- **Intersect** Shape Parts
- Less-Destructive DXF, Split, Merge (editable rectangles where possible)
- **Simplified Export** to VisualAnalysis (operation & layout, some properties predefined)
- Partial-Containment Selection (CAD like box) is more accurate
- **Change database part-size** after dropping (within the category)
- Dragging by Vertex is much more accurate
- Export aluminum shapes to VA 12.0 database works properly

User Interface/Usability

- Better editing of multiple selected items
- Simplified **dimensioning 'mode'** with a cursor
- **Preference Settings** have their own Dialog, better organized
- Color Preferences for many graphical items
- **Improved Graphics** (colors and sizes, selection and highlight, drag & snap)
- Modify annotations or dimensions in Project Manager
- Easily edit **multiple-line text annotations**
- Modify '**project settings**' in Project Manager
- **Pan** the Graphical View with Arrow-Keys
- View Tabs moved to top for easier access and visibility
- Undo in"Quick Access" (above the ribbon/menu)
- "Settings", like preferences but automatic for various options
- **Improved rebar information** in R/C shapes reports

Quality Control/Performance

- Behind the scenes quality-control features for validation
- Improved crash-handling
- Updated to latest compilers & tools
- Crash-prevention work

Architecture

- 64-bit multi-threaded architecture (requires 64-bit Windows)

Version 8

Introduction

ShapeBuilder 8.0 offers an incremental improvement over version 7.0 with the following improvements to streamline your work flow

Major Features

- **Explode Array** command allows you to create asymmetric arrays or remove some parts from an array.
- **Override Shape Color**: see parts more clearly by optionally overriding material colors
- **Quick Rotate**: button on toolbar will rotate a selected shape by a preference-setting angle
- **Unit Style Selection**: easier access on the toolbar
- **Material Search**: dialog box offers a **search** feature, and other minor improvements.
- **Copy Command**: extended to place both a picture and all results (tabbed-delimited) on the Windows Clipboard
- **Export to DXF**: Shape outlines (and holes) will export to a CAD-compatible DXF file.
- **Streamlined User Interface**: layout is less distracting to your work
- **Custom Report Logo**: Add your own company logo to the report header, see Reports for details

Minor

- **Default materials** are 'remembered' as preferences for your next project
- Upgraded to **CFS 8.0** implementation for cold-formed .scl import
- **Simplified** Export to IES Database dialog box

Version 7

Introduction

ShapeBuilder 7.0 represents a **complete rewrite** of the software from the inside out. User's of previous versions should brace themselves for some fundamental (and excellent) changes. We hope you enjoy the fruit of our labor. The new system is built on the latest technologies such as multi-threaded operations to leverage the power of the processor in your machine. We have streamlined the steps required to work with shapes. We have removed requirements to destructively modify shapes before certain operations. We have streamlined and overhauled the user-interface to reflect actual usage of the product. The new system uses the concept of a "Flattener" which looks at overlapping parts of the same material and flattens them into a single boundary. Any hole shapes are then subtracted from this. It is important for you to use the "F" key in the Sketch View or the Filter option to show the flattened shape and verify it is what you expect it to be.

Engineering Improvements

- Updated all Shape and Material **databases** from AISC, NDS, AISI, ADM, etc.
- Improved **Meshing** for finite element analysis: "nice elements" and easy to control
- Improved **accuracy** in a dozen different ways for property calculations
- **Rebar** selection, arrangement, and adjustments are far easier in reinforced concrete sections
- **Parametric shapes** are more completely and consistently described
- Parametric shape "part properties" are available, similar to database shapes
- **Advanced analysis is smarter** about when it is applicable
- **Monosymmetry factor** "Beta 1" added to advanced analysis
- Improved DXF file import capabilities: No need to "Simplify" complex shapes!
- New **import geometry** formats available: BREP, CSFDB

Editing Improvements

- **Holes** are incredibly easy and flexible: toggle **any shape** to make it a hole
- **Notch** a shape with partially overlapping hole.
- **Modify multiple shapes simultaneously** to set common properties, or to align positions
- **Snap Side-By-Side**: Easily snap two selected shapes side-by-side or top-to-bottom with a toolbar button
- **Snap Vertices**: Drag one shape by a vertex and snap it easily to any other shape's vertex.
- **Constrained dragging** of shape vertex (Ctrl+) to stop when a shape 'hits' another
- **Shape "Arrays"** let you quickly and easily modify a rectangular or circular pattern of shapes
- **Start Screen** makes it easy to pick up where you left off or to start a new project
- **Shape rotations** are much easier to understand, rotation angles persist
- Alignment with a **Locked** Shape (other parts will align to the locked shape)

General and Performance

- **Multi-threaded architecture** to take advantage of modern processor capabilities

- FEA Meshing is 1000 times faster
- Faster loading of databases on startup
- Select general or reinforced concrete project at the start for a simpler, smarter interface
- Advanced analysis is much faster and automatically run in the background
- Built-in crash-handler to report detailed diagnostics to IES for improved quality
- Changed graphics from OpenGL to WPF/DirectX for improved performance and reliability
- Get a **shape preview** for the .SBF file in the Startup dialog
- Better error handling and presentation

Features Removed or Changed Significantly

Advanced torsional analysis of "separated" shapes or composite materials

We have removed the ability to calculate J for "disconnected" shapes or composite-material shapes. The reason we removed it, is because the entire calculation is based on the Warping Normal function and in the past we were making some very loose assumptions that we are no longer comfortable with making. How do two "independent" shapes behave in torsion? Does super-position apply? We don't have any good theoretical or test-data answers to this question, certainly not in the general sense for all shapes. You could analyze each part independently and use superposition if you think it is appropriate. You could provide some kind of connector parts to indicate how these shapes will be tied together during the torsional loading. Or you could help us track down some kind of published information on how to solve this problem!

Features Changed:

- Ability to edit the IES Shape database (files are user-friendly now, edit outside ShapeBuilder)
- Copy properties (as text) to Clipboard (new method: Right-click on Results, Select All, Copy)
- Paste outline data from clipboard to import shapes (redundant, use File | Import)
- Part Properties *in reports* (except Q, but still available in the Modify tab)

Features Removed:

- Embedding rebar or holes is no longer required
- "Simplify" complex shapes, usually DXF imports. (for functionality & performance problems, not needed)
- Export to DXF file (not used)
- Import centerline coordinates (not used)
- Drawing/Editing Grid (useless)
- Copy image to Clipboard (just use Alt+PrtScn, crop away in Paint, Word, Excel...)
- Shape linking (not useful?)
- Shape Intersect, Inverse Intersect (not needed for anything)
- Automatic Load Annotation (not useful)
- Stress Graphs in the result view (not used)
- Annotation leader lines (not needed)
- Many Preference Settings (most were not needed or used)

Defects Corrected

- Composite shape plastic section properties were not correct

Version 6

Editing Improvements

- Added **Isosceles triangle** parametric shape for easier shape creation.
- Shapes no longer "jump" (move) when selecting a vertex.
- **Selection** of shapes and sub-shapes is easier to see.
- Selection of **Rebar** is Easier, more Visible
- Improved **rotation** of shapes, linked shapes, shapes with holes and sub-shapes
- **Project Properties** in Modify tab, simplifies Project Manager
- Improved reliability and error-checking for importing shapes (DXF, STEP, etc.)
- Ability to export custom **aluminum shapes** to VisualAnalysis 8.0+ for design checks
- Easier editing of **dimensions and annotations**

Analysis Improvements

- **Improved FEA Mesher**, significantly faster and more robust
- Advanced Stress Analysis is much **more accurate** for multi-part shapes
- Advanced Stress Analysis is **significantly faster** than previous versions
- Click on Shape in Result View for **Stress Graphs** for better result visualization
- Sheer stress colors are consistent among all result windows, easier to understand
- **Load Locations** and Values are in an Automatic Annotation in Result Views
- **Concrete Analysis** reports Mo for both directions (Mox, Moy)
- Reinforced Concrete Analysis is more robust.

Other Improvements

- Right-Click on Shape Database to "**Open File Location**" (shows your .dbs files in Explorer)
- Improved importing of CFS .SCL files for cold-formed shapes
- Semi-automatic "simplification" and clean-up of imported shapes outlines.
- Fixed problems in **Preferences** for material defaults
- Improved tolerances for working with very small shapes.
- Some new preference settings were added.
- Simplified shape tree for parametric shapes (less clicking to find shapes).
- Advertise ability to create arbitrary reinforced concrete shapes, with instructions
- Direct menu link for training videos or tutorials
- Error-prevention code added.
- Numerous minor bug-fixes.

Features Removed or Modified in 6.0

- **Analysis** tab in Project Manager is now the Advanced Settings tab.
- *Export to VA 5.5* shape database option
- **Effective Section Analysis** calculation (very limited, did not work well, and was not used)
- General triangle shape (as unworkable!)
- **Simplify Complex Shape**: replaced by more automatic system on shape import. (If you need to re-simplify a shape to solve analysis problems it is best done by starting with the original shape again.)

Version 5

New Features

- **Mouse-wheel** support for zoom/pan
- Much Improved **DXF Import** (loose lines & polylines)
- Import and Export IGES / STEP files
- Easier **import of custom shape libraries** through the File Menu
- Calculation of **First Moment of Area**, Q for parts
- Calculation of **Shear Flow** (VQ/I) at part boundaries
- **Angle** Dimensions (degrees)
- Report **Printing Options**
- **Leader-lines** for Annotations (optional)
- Graphics may be copied to Clipboard
- Added **Orientation** setting for Parametric Shapes
- **Torsion constant** is calculated without advanced analysis for more shapes
- Include **Part Results** in Printed Reports
- Easier Selection of Annotations
- **Simplify Complex Shape**, allows advanced analysis of really complex shapes
- Shape Database Overhaul
    - Add/Edit/Delete functions are now available for custom databases, categories, and shapes
    - Custom database shapes can be imported from or exported to the clipboard
    - Improved database loading performance
    - Shape database re-write improves over all program performance
- Advanced Analysis
    - Advanced Analysis tab in project manager for more visible access
    - Improved Advanced Analysis performance, meshing & accuracy
    - Improved feedback and messaging for advanced analysis
- Numerous **bug fixes** to improve accuracy, usability and reliability
- Expanded and improved help and tutorials

Version 4.5

New Features

- Database Operations (formerly only in the Shape Database Editor utility)
    - Import Custom Shapes directly
    - Import a CFS library of cold-formed steel shapes (.scl file)
    - Create, Rename, Delete shapes and categories
    - Edit database shape properties
- Calculate Cracked Moments of Inertia in concrete analysis
- Works with VisualAnalysis 6.0 data files.
- Easier export of shapes for use in VisualAnalysis

    - Ability to export multi-part shapes to VisualAnalysis(merge requirement waived)
    - Smarter export of wood shapes to VisualAnalysis (correct property sets defined)
- Easier, simpler unit system
- Shift+Tab selects shapes in reverse order, and Edit | Select Previous shape command.
- Pipe sector shapes can be positioned or dimensioned from the center point of the arc.
- DXF file import improvements
- Reinforced concrete "Wizard" now functions correctly with metric bars.
- Easier to use "Text Label" feature
- Visual feedback on "Snapping" shapes by vertices or side-points
- Correct calculation for shear center location with multi-part (not merged) shapes.
- Easier import of data for shape outlines & centerlines, flexible units.
- Merge multiple shapes simultaneously
- Advanced Loads are saved with project file.
- No "jumping" when dragging shape dimensions.
- Some performance improvements
- Some stability improvements
- Minor report improvements
- Part data included in Clipboard (text report)

Version 4

- AISI Effective Section Properties Analysis
- Easier Export to VisualAnalysis
- Advanced Stress Analysis for Non-Merged Shape Groups
- Units and Layout Saved with File
- Resultant Shear Stresses Calculated and Plotted
- Report total shear stress
- Adjustable report margins
- Adjustable report fonts
- "Flyby Tips" for stress values in result views
- Reposition legends in result views
- Create dimensions for concrete results
- Advanced stress analysis for shapes with "notches" (holes on the edge)
- DXF file import and export improvements
- Import a centerline or outline shape based on "x-y points"
- Yield stress-weighted plastic modulus for composite shapes
- Mid-point snapping (also 1/3 points, 1/4 points etc.)
- Improved dimension editing
- Lock Shapes in place to prevent accidental movement
- Hole creation based on shape intersection
- Polar grid in sketch view, for radial editing and snapping
- "Mouse-wheel" and "Area" zooming
- "Mouse-wheel" scroll/pan

- Drag box selection by simply dragging mouse
- Property calculations no longer slow down editing
- Single-step "Mirror & Copy" command
- Mirror multiple shapes about a common point
- New parametric shapes
- Improved Preference settings
- AISI Effective Section Properties Analysis
- Easier Export to VisualAnalysis
- Advanced Stress Analysis for Non-Merged Shape Groups
- Units and Layout Saved with File
- Resultant Shear Stresses Calculated and Plotted
- Report total shear stress
- Adjustable report margins
- Adjustable report fonts
- "Flyby Tips" for stress values in result views
- Reposition legends in result views
- Create dimensions for concrete results
- Advanced stress analysis for shapes with "notches" (holes on the edge)
- DXF file import and export improvements
- Import a centerline or outline shape based on "x-y points"
- Yield stress-weighted plastic modulus for composite shapes
- Mid-point snapping (also 1/3 points, 1/4 points etc.)
- Improved dimension editing
- Lock Shapes in place to prevent accidental movement
- Hole creation based on shape intersection
- Polar grid in sketch view, for radial editing and snapping
- "Mouse-wheel" and "Area" zooming
- "Mouse-wheel" scroll/pan
- Drag box selection by simply dragging mouse
- Property calculations no longer slow down editing
- Single-step "Mirror & Copy" command
- Mirror multiple shapes about a common point
- New parametric shapes
- Improved Preference settings

Version 3

ShapeBuilder 3.0 is built on a whole new series of technology that will be appearing in future IES products. Some of the highlights of the new version include:

- Better Editing, Alignment, and Scrolling
- Better Export to VisualAnalysis
- Export for use in VisualDesign
- Easier Dimensioning and Annotating

- More Properties Calculated
- Better Printed Reports
- Draw Centerline of Thin-Walled Shapes
- Sketch Arbitrary Outlines and Holes
- More Parametric Shapes
- Import Shapes from DXF Files
- Advanced FEM Stress and Torsion Analysis
- Advanced Reinforced Concrete Analysis
- Expanded Shape and Material Databases
- Component Part Properties are Calculated
- Smart, Friendly Look and Feel

1.6 Preferences

There are several default settings that you may adjust to make ShapeBuilder look and feel more like you desire. You may explore these settings to adjust operations, filters, graphics, colors, fonts, and reporting. Some settings do not take effect until you restart the program or start a new project. 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 History Files

We create copies of a project-file when it is saved. These are stored in the *C:\Users\<your.login>\AppData\Local\IES\Customer\HistoryProjects* folder. If you have data corruption or lost a project, you might be able to recover from one of these. The number of days these files are kept is set in the Project section of the preferences. You can turn this feature off (to save disk space or if using your own backup mechanisms) by setting the value to zero.

Project, Next Inspector Field on Enter

By default, in the Project Manager, Modify area, when you press the Enter key it sets your input data but does not change focus to the next entry field. The Tab key lets you move from one field to another. You can enable the movement to the next field on the Enter key with this option.

Reports, Customer Logo

You can use your own company logo in text reports (Report Viewer). Ideally you'll use a .bmp, .jpeg, or .png that is a reasonable shape and size. Most printers print at 300 dpi, so a 2" logo might be 600 pixels square. ShapeBuilder will scale your image to fit the header area of the report. You can use a logo anywhere on your machine, but if you drop one in the *C:\ProgramData\IES\Customer\ReportLogo.jpg* location, we'll find it automatically.

1.7 Support Resources

Did you Search this Help File?

Be sure you make use of the help and support built into the software, as described in the Program Layout section of the

User's Guide. This document may be searched, and you should try various search terms, sometimes less is more when searching -- use just the unique word or words. There is also a logical Table of Contents available.

Do Not Contact Support For

- **Licensing** or **Sales**: use www.iesweb.com/service or sales@iesweb.com.
- Questions about **how to model** a particular structure. Such questions are your responsibility as an engineer.
- IES cannot **validate your model** or your results. If they "seem" incorrect, please figure out WHY they are incorrect. If you can document a defect, we will be happy to investigate deeper and fix things as necessary, but we cannot afford to check every customer's model.
- Questions about **engineering theory**. IES is not in the business of educating engineers. There are textbooks referenced in this help file and we can provide more guidance as to where to look if you cannot find one.

Technical Support

- **Email Support:** support@iesweb.com (Replies are usually **within 2 business hours**, if you don't hear anything within a 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 or relevant, including attaching a project file. "*I have a problem, can you help?*" is a frequently submitted question, to which the answer is always: "*Maybe, what is the problem?*".
- **Telephone Support:** No, sorry. 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: sales@iesweb.com.
- **Free Training Videos**: Training videos can be accessed here.

2 Modeling

2.1 Project Settings

Access the project settings by going to **Project Manager | Modify** when nothing is selected. Click on the background of the Sketch View to unselect everything.

Base Material

When your shape contains multiple parts of different materials, you can choose which material to use as the base, or denominator for the modular transformed section properties. By default, the first shape you create sets this base material.

Shape Flattening

This setting allows you to take charge of how to process the shape parts to construct the final shape for analysis. The flattened model is shown in the Analysis View. Examine the flattened shape to ensure that the shape is analyzed as expected.

Auto: Let ShapeBuilder flatten the model. ShapeBuilder will try to correctly merge touching parts, remove overlaps, and subtract holes. Shape parts (including holes) may overlap and holes may be used to create notches.

Manual: You are responsible for flattening the model. This setting allows you to 'take control' of flattening if the software is not producing the model that you want. In the manual mode, you are required to remove overlaps and to merge touching shapes. This option is useful when trying to model a shape part that is completely contained within a hole, for example.

Mesh Options

You may control the automated finite element mesh generator for situations where the artificial intelligence gets confused by the special geometry of your shapes. You can improve performance of analysis by adjusting these settings to create a more intelligent element distribution in the shapes.

2.2 Shape Types

There are three types of shapes available for use in ShapeBuilder: generic section parts, parametric shapes, and database shapes.

Generic Section Parts

Generic section parts are created by either drawing the part in the Sketch View or by importing the parts from a CAD or text file. Any arbitrary geometry can be constructed by drawing multiple parts connected to or overlapping one another. Once a polygon is drawn or imported, the dimensions of the part cannot be modified.

Parametric Shapes

Parametric shapes are defined by dimensions like width, depth, radius, thickness, etc. that can be changed manually. Parametric shapes can be used as building blocks to create more complex custom shapes. ShapeBuilder offers 34 types of parametric shapes to choose from which fall into three categories: Simple Parametric, Rolled Parametric, and Bent Parametric.

Database Shapes

ShapeBuilder has two different types of shapes databases: a standard shape database and a custom shape database. Custom shapes created in ShapeBuilder can be added to the custom shape databases. IES VisualAnalysis can perform design checks on member elements created from shapes in the IES shape database as well as some certain shapes Created in or Exported from ShapeBuilder.

Common Shape Databases

IES includes a large database of steel, wood, cold-formed, aluminum, and other shapes common to the building industry (e.g. AISC, ACI, NDS, etc.). While the dimensions of these shapes cannot be modified in ShapeBuilder, these shapes can be combined (e.g. a cover plate can be added to a wide flange) and saved in the custom shape database. 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. While, you may use these shapes, please understand their purpose and limitations before using them.

Custom Shape Databases

Custom Shape Databases are XML files which store custom shape data that is used by various IES programs to defined the cross-sections for member elements. For convince, a Custom Shape Database can be added, edited, and deleted directly in ShapeBuilder as outlined in the Database Customization section of the help file and as discussed in the Database Customization training video. Additionally, a Custom Shape Database can be generated by Exporting shapes to the IES Shape Database. While often not necessary, Custom Shape Databases can be modified manually outside of ShapeBuilder and there are examples files in the Shapes folder for convenience. Note: The program must be restarted to load any database changes. Custom Shape Databases are stored at following path: *C:\Users\<your.login>\AppData\Local\IES\Customer\Shapes* which is easily accessed by clicking the ***Custom Data*** button in the **Home** ribbon. To share a Custom Shape Databases, simply copy the XML file to the Shapes folder on another machine.

Cold Formed Shape Library

To get custom cold-formed steel shapes into the IES Database for ShapeBuilder or VisualAnalysis, you may use the menu item **File | Import | Import Cold Formed Library** to import the .scl or .cfsl file you have obtained through CFS. VisualAnalysis has the ability to design or check cold-formed steel shapes; however, these shapes must come from RSG Software's CFS program (www.rsgsoftware.com), because VisualAnalysis uses that utility, behind the scenes, to perform the design checks.

2.3 Database Customization

Custom Shape Databases are XML files which store custom shape data that is used by various IES programs to defined the cross-sections for member elements. For convince, Custom Shape Databases can be added, edited, and deleted directly in ShapeBuilder as outlined below and as discussed in the Database Customization training video.

Database

To add a new Custom Shape Database, right-click in the white space of the **Project Manager | Create** tab and select ***Add Database***. The name and image path for the database are defined in the dialog box that appears. Right-click on an existing Custom Shape Database to edit or delete the database or to add a category to the database.

Category

To add a new Shape Category to a Custom Shape Database, right-click on the database and select ***Add Category***. The

shape type, name, and image path for the category are defined in the dialog box that appears. Note: The shapes in a category can only contain a single shape type (e.g. both I-shapes and channels cannot exist in the same category). Right-click on an existing category to edit or delete the category or to add a shape to the category.

Shape

To add a new Shape to a custom shape database category, right-click on the category and select ***Add Shape***. The name, default material, dimensions, and cross-section properties can be set in the dialog box that appears. Note: The cross-section properties can be entered manually or the Calculate Properties feature can be use to automatically generate all possible section properties from the specified dimensions. Right-click on an existing shape to edit or delete the shape.

Clipboard Import / Export

To export all of the shapes in a category, right click on the category and select ***Export Shapes to Clipboard***. The shape information can then be pasted into a spreadsheet or text file. Note: Even if the shape category is empty, the ***Export Shape to Clipboard*** option will copy the header information to the clipboard for the shape type of the category. This makes it easy to learn the format for a particular shape type that should be used for the ***Import Shapes from Clipboard*** feature. Note: When shapes are imported from the clipboard, some of the section properties are generated automatically. Right-click on an imported shape to manually adjust the section properties as needed.

2.4 Shape Operations

Creating Shapes

There are three ways to create shapes in ShapeBuilder: draw generic section parts, select parametric or database shapes, and import shapes from a file. Watch the Creating & Modifying Shapes video for an example of how to create shapes.

Drawing Generic Parts

Arbitrary polygon parts can be drawn in the Sketch View by selecting **Home | Draw Shape** from the Ribbon. A drawing grid is typically defined before sketching, but a polygon can be drawn to existing vertex points. Parts have "snap" points and the snap distance can be specified in the **Project Manager | Filter** tab. Any arbitrary geometry can be constructed by drawing multiple parts connected to or overlapping one another. Once a polygon is drawn, the dimensions of the part cannot be modified.

Selecting Parametric or Database Shapes

To use a shape from the database, simply ***Drag*** a shape from the **Project Manager | Sketch View** with the mouse and position it as needed. Alternatively, double click on the shape to have it appear in the sketch view with its center of gravity located at the origin.

Import Shape Geometry from CAD Outline

The ability to import DXF, DWG, IGES, STEP, BREP, and CSFDB files into ShapeBuilder provides a direct link to your favorite CAD package. ShapeBuilder cannot import all DXF files, and IES does not control these file formats. Polygons contained in these files do not have materials and are not "nested" or otherwise related. Each polygon is imported independently, and those with a clockwise ordering are assumed to be holes. Each polygon will become an independent "part" in ShapeBuilder. The data is assumed to be in the unit system you choose when prompted by ShapeBuilder. Watch the Importing & Exporting video for an example of how to import shapes from CAD.

You should simplify any DXF file down to purely Model Space objects (just the outlines of the shapes). The DXF file is assumed to contain one or more shape outlines defined as Polyline entities. If your DXF file contains disjointed Line and

Arc entities, ShapeBuilder will attempt to create a "closed chain" outline from them, but it may not work. Other entities in the file may or may not be imported properly. It is always recommended that you verify the model is correct after importing into ShapeBuilder.

Import Shape Geometry using a Text File

You can create a shape by describing the outline using x,y coordinate points in a text file. This is useful if you have a complex shape that is not handled by the built-in Parametric Shapes. Simply list the data points in a text file (.txt) that is space, tab, or comma delimited. Here is an example for the outline of an L shape:

cm
0.0, 0.0
2.0, 0.0
2.0, 0.25
0.5, 0.25
0.5, 3.75
0.0, 3.75
0.0, 0.0

The units line is optional; inches are assumed if one of the following is not specified: {in, ft, yd, mm, cm, m}. You do not need to list the first point twice, the shape will be "closed" automatically. Imported shape sizes cannot be modified after import, unlike parametric shapes created within ShapeBuilder. Watch the Importing & Exporting video for an example of how to import shapes from a text file.

Modifying Shapes

When part or parts are selected, their properties can be changed in the **Project Manager | Modify** tab. Below are the properties that can be modified many of which are also covered in the training videos.

Material Properties

Each part in your assembled shape is made of a specific material. If all your parts have the same material, your shape is non-composite. For composite shapes you must specify one Base Material, which is a material used in one of the parts.

Holes

Also, any part can be turned into a hole by setting its "*Is a Hole*" property to true in the **Project Manager | Modify** tab when the part is selected. Holes may be fully contained or may overlap the boundaries of other shapes. Anything "under" the hole part is subtracted. Complex holes can be created out of multiple hole parts. Switch to the Analysis View to see if the shape has been correctly interpreted for analysis. Holes are handled differently between the two Shape Flattening modes. With the Auto mode, you can overlap parts, including holes, to create notch effects. With the Manual mode, holes and other parts must be fully contained or simply touching other parts. See Project Settings for more information on Shape Flattening.

Lock

This command disables many of the options in the Modify tab such as: hole, rotation, dimensions, location, etc. This is useful if you want to fix the location of some part(s) so you don't accidentally move it/them while modifying other parts. If you are trying to arrange lots of small parts around one main part, locking the main part can make it easier.

Rotate

Entering a positive rotation angle causes the selected part(s) to rotate about the part's centroid in a counter-clockwise

direction. More rotation options (Quick Rotate Shape and Rotate about a Point) are available in the Shapes tab of the Ribbon.

Mirror and Flip

The Mirror command causes the part to be mirrored about the vertical axis without changing the shapes rotation angle. The Flip command mirrors the part and increases the rotation angle by 180-degrees.

Dimensions

Only the dimensions of parametric shapes can be changed (the dimensions of database or imported parts cannot modified, except by overlapping holes or Merging with other parts). Clicking the Definition option in the **Project Manager | Modify** tab will display of graphic of the part with the dimensions defined.

Location

You can specify the exact location of your shape by entering in values for the Centroid, Left, Right, Center, or Polar Center. You may use mathematical expressions in these boxes to add or subtract a specific distance from the current location. You can also move parts using mouse or keyboard commands or using the Move commands in the **Ribbon | Shapes** tab.

Arrays of Parts

The Array command automatically generates an array of the part(s) selected. The number of parts and the spacing of the parts can be modified and the arrangement of the array can be changed from rectangular to polar. Parametric shapes remain editable (size) while arrayed. The Explode Shape Array command from the **Ribbon | Shapes** tab can be used to turn the array into several independent parts.

Split a Shape

The Split command (from the **Ribbon | Shapes** tab) allows you to slice a shape into two parts either horizontally or vertically at the centroid or a specified location. The resulting shapes, unless they are rectangular, will not be parametric shapes with editable dimensions.

Arranging and Modifying Multiple Shapes

Some commands are only valid when more than one part is selected. All of these commands are located in the **Ribbon | Shapes** tab and are described below. While the merge, intersect, and subtract commands can be used they are typically not required as ShapeBuilder automatically performs these operations when it "flattens" the shape. Watch the Shape Flattening video for more on this topic.

Merge

The Merge command combines multiple shapes into a single shape. If you merge parametric shapes, you will no longer be able to edit their dimensions. Therefore, you may wish to save a copy of your project for editing before merging shapes. For this command to be available the shapes must be touching and all shapes to be merged need to be selected. You cannot merge shapes with different materials. You can merge a shape with holes to create a part reduced by the holes (this will cause the original holes to vanish).

Intersect

This command creates new shapes from the intersection of two (and only two) parts. Intersecting complex parts can result in multiple parts or holes. Only the resulting shapes that are rectangular will be parametric shapes with editable

dimensions.

Subtract

You can subtract overlapping holes from a part to create the net piece. If the original is parametric, you may lose the ability to edit the dimensions.

Snap

The Snap commands move two shapes together so that they touch or align left-to-right or top-to-bottom. You can also snap a shape's vertex to any other shape's vertex, by dragging one vertex towards the other. As you get close (within 1/8th of an inch) you will see your shape jump to that point and a large circle will flash to indicate that they have snapped.

Aligning Shapes

A variety of alignment and spacing commands are available so that you can easily position your parts relative to each other.

2.5 Material Properties

Material properties for the shapes come from the IES material database, which includes most typical materials you might need. To change a material, select a part use the **[...]** button next to "material" in the **Project Manager | Modify** tab, to open the material database dialog. After selecting Project Settings in the **Modify** tab, you can select the Base Material for your project. All materials in ShapeBuilder are linear, elastic, and isotropic. If you cannot find the material you need, you can add a custom material to the database.

Custom Materials

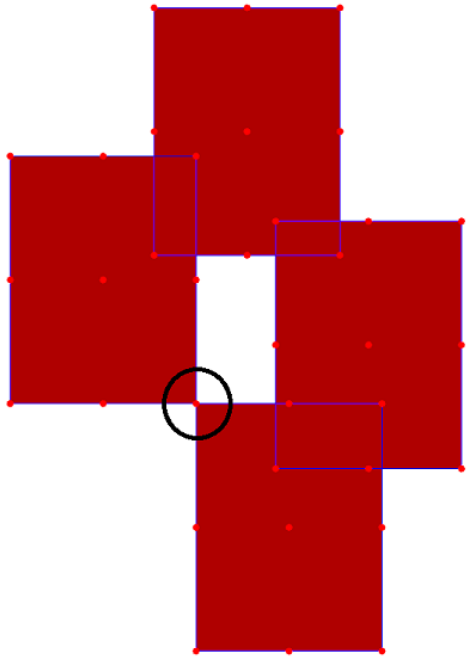
You can add custom materials to your system by clicking the **Add Custom Material** button in the Material Database dialog box. The Material Database dialog box can also be used to edit or delete a custom material by right clicking on the material. While only custom materials can be edited or removed, any material can be cloned as a custom material by right-clicking on the material. When defining a material, you can specify the material "type" making it possible to define custom materials that will be categorized as wood, steel, concrete etc. After selecting a material type, the defining properties have default values that can be modified for your custom material. If you do not know all of the defining properties for your material type, consider using a General material type where you only need to define four basic material properties: modulus of elasticity (E), Poisson's ratio (nu, ν), the weight density (gamma, γ), and the coefficient of thermal expansion (alpha, α). The Shear Modulus, G, is calculated internally as: G = E / (2*(1 + nu)). These materials are available in all IES products.

Custom Material Database

The Custom Material Database is a set of XML files used to manage the custom materials in the IES products. The XML files are stored on your machine at the following path: *C:\Users\<your.login>\AppData\Local\IES\Customer\Materials*. You may copy these files from one machine to another to share your customizations. While you can modify the database files manually outside of ShapeBuilder, this is generally not needed as you can add, remove, edit, and clone the custom materials in the Material Database dialog box as explained above. If you do decided to manually modify the database files, there are examples in the Materials folder for you to follow. You will need to restart the program to load any database changes.

2.6 Simple Polygons

ShapeBuilder will only analyze simple polygons. A simple polygon is one that does not self-intersect, or have touching vertices. Vertices touch if two points share the same location (and are not adjacent). The screen shot below shows a polygon for which analysis fails. The merged outside boundary is not a simple polygon, due to the shared vertex between the rectangles, which encloses the hole (circled in black). Less obvious complex polygons are sometimes created by small misalignments in the shape's geometry (e.g. small gaps where shapes are meant to connect). To fix the analysis problem, either increase the overlap so that it is more than a point or create a gap so that the shape is not closed.



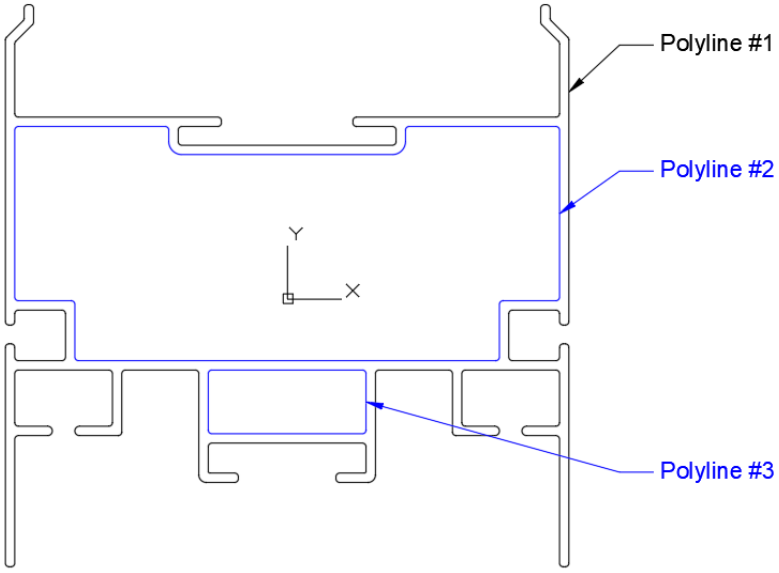
2.7 Example: Import DXF Shape

Step 1: Create DXF/DWG File

Using a CAD program, create a .dxf or .dwg file of the cross-section you would like to Import into ShapeBuilder. The section should be composed of Polyline, 2D Polyline, 3D Polyline, Circle, Ellipse, 3D Face, or Solid objects.

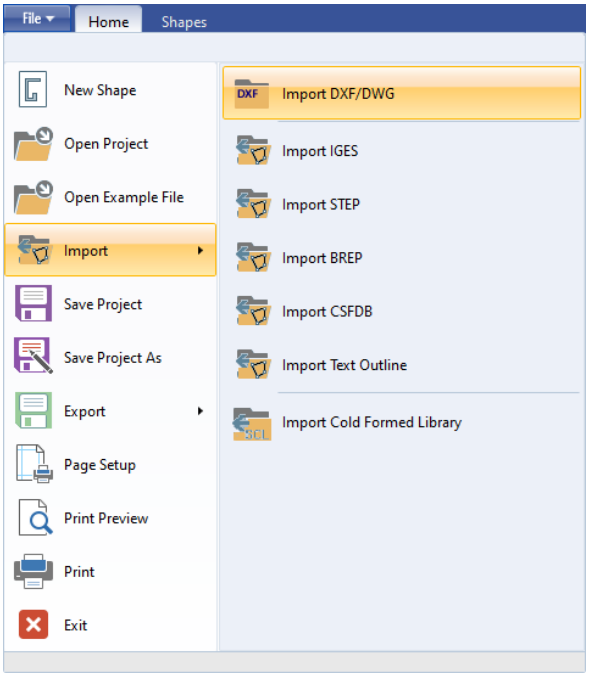
When using Polyline objects, it is expected that the polylines form a closed loop in a single plane. If the object being imported lies off of the Global XY plane, an attempt will be made to "squash" the object onto the XY plane.

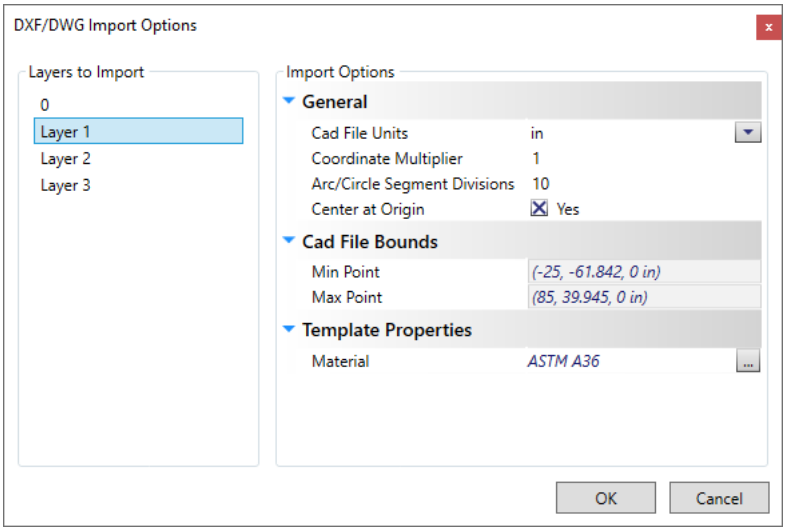
In the figure below, Polyline #1 is the shape outline and Polyline #2 and #3 are contained inside of #1 and will be turned into Holes within ShapeBuilder.



Step 2: Import Settings

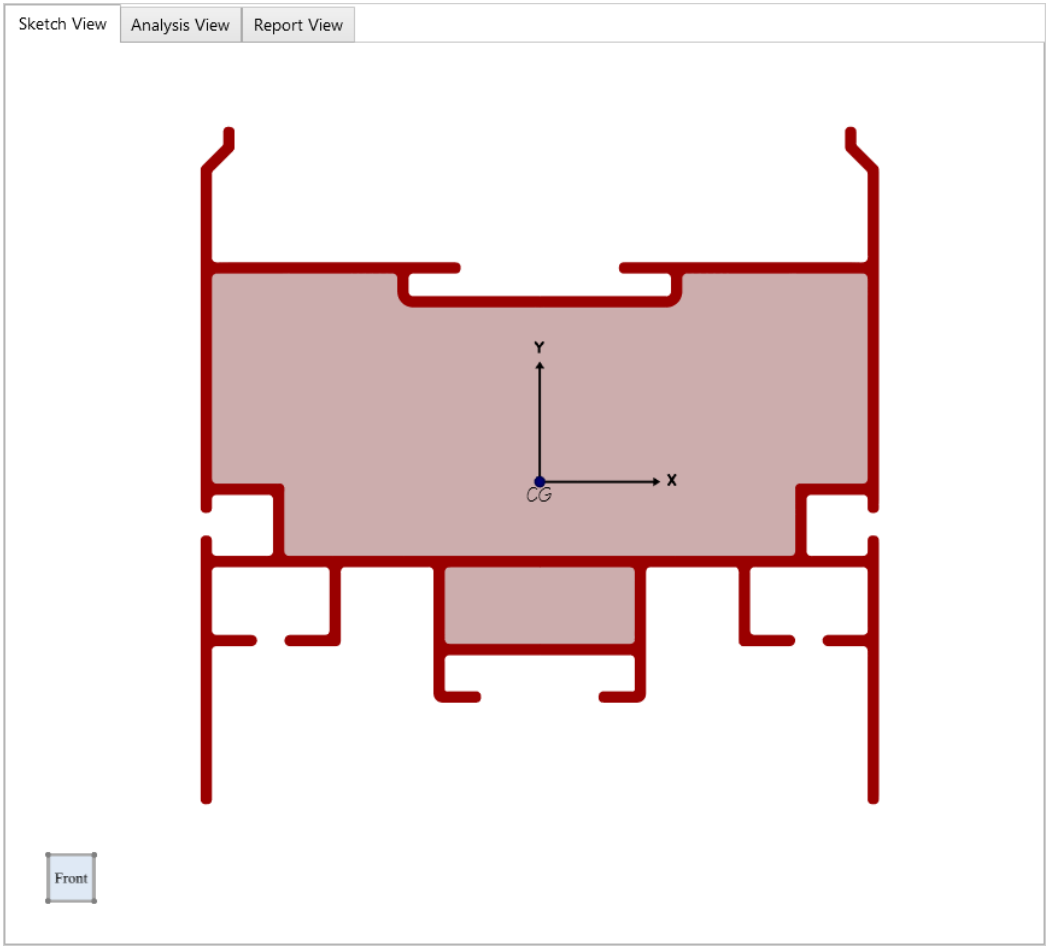
From the File menu, select **Import | Import DXF/DWG** and browse to the file created in Step 1. This will bring up the DXF/DWG Import Options dialog, where you can select the Layers to Import and set other miscellaneous options for the import process. Once everything is set properly, select OK to Import the file. Note: Tool tips provide additional information by hovering your mouse over each field, including the Layers available to import. The Material field defaults to the User Preference 'Default Parametric Material' setting.





Step 3: Resulting Shape

The image below shows the resulting shape that has been Imported into ShapeBuilder.


Sketch View
Analysis View
Report View
Y
X
CG
Front

3 Analysis

3.1 Analysis Overview

The various calculations performed by ShapeBuilder fall into two categories: Simple Analysis and Advanced Analysis. The Simple Analysis calculates properties using numeric integration while the Advanced analysis results rely on the Finite Element Mesh. Depending on the characteristics of the shape, there are certain instances where some calculations are not supported. For example, when a shape has two or more disconnected parts (i.e. Multiple Boundaries) and/or two or more material properties (i.e. Composite), the Torsion Properties and the Shear Stress Results are not calculated. The table below shows when each set of calculations is supported and on what the calculations depend (i.e. Finite Element Analysis, Finite Element Mesh, and Loads). In the Analysis Settings, the Finite Element Mesh can be refined and the Loads Applied. Watch the Analysis Results video for an overview of the results that are available in ShapeBuilder for various models and loading conditions.

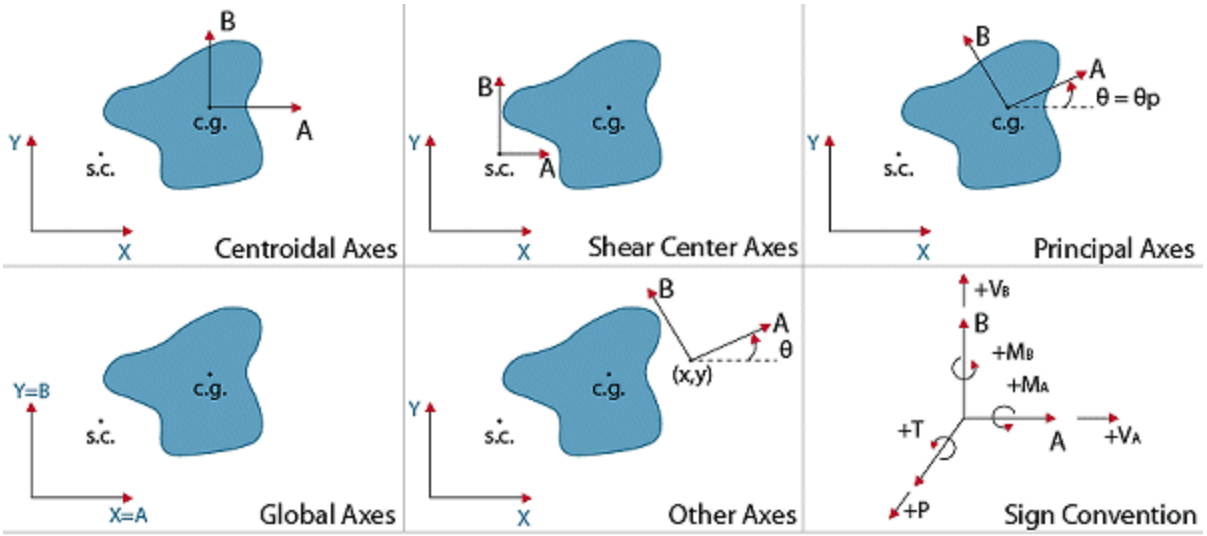
Overview	Multiple Boundaries / Composite	Finite Element Analysis Dependent	Finite Element Mesh Dependent	Load Dependent
Simple Analysis				
Overall Geometric Properties Principal Properties Polar Properties Plastic Properties Part Properties	Supported	No	No	No
Advanced Analysis				
Normal Stresses	Supported	No	Yes	Yes
Torsion Properties	Not Supported	Yes	Yes	No
Shear Stress Results	Not Supported	Yes	Yes	Yes
Warping Function	Not Supported	Yes	Yes	No
Shear Flow	Supported	No	Yes	Yes

3.2 Analysis Settings

Finite element analysis is used to determine the torsion properties and the shear stresses for the shape. Mesh refinement is required to get accurate results. The analysis runs automatically in the background and you do not need to wait for analysis to finish when editing your shape.

Input

The **Analysis Settings** tab in **Project Manager** (when the Analysis View is active) allows input for the applied loading on the section, the location of the applied loading (body forces), a mesh refinement adjustment, and a selector to determine which of the available results to display in the graphic window. Shown in the figure below are the orientations and positive sign convention for the applied loads on the section.



Note: Use tool-tips on individual inspector properties for more information

Coordinate System

The location of the applied load is specified in the drop down box. The options here include: the Global Axes (X,Y), the Centroidal Axes (x,y), the Principal Axes (1,2,theta), Shear Center, and Other Axes. If Other Axes is chosen, you will need to specify the X and Y coordinates (in the global system) of the origin of your coordinate system as well as the angle of rotation. The location you specify can cause implicit eccentricity as described below.

Applied Loads

Load	Description
Axial Force, *P*	An applied axial force, where compression is negative, tension is positive. If the load is applied at a point other than the centroid, it will also induce moments on the cross section.
Bending Moments, M_a, M_b	Moment sign conventions are defined in the figure above.
Shear Force, V_a, V_b	Positive shear is in the direction of the positive axis. Shear forces applied at a point other than the shear center will induce torsion as well.
Torque, *T*	Applied torsional moment, counterclockwise is positive, following the right-hand-rule.

Loads are applied as body forces not concentrated forces. Since they are applied as body forces, stress concentrations are not considered.

Finite Element Mesh Refinement

The mesh refinement slider is used to control how many elements are used in the analysis. You can compare the results

between different meshes to determine if the results are "converging" on the true elasticity solution. Finite element analysis is inherently approximate. To ensure your analysis results are accurate you should follow a mesh refinement technique, which is outlined below. The goal is to get accurate results, while minimizing the time you spend waiting for results. Generally, you use a coarse mesh to get fast preliminary results, and then use finer meshes for final results.

1. You must run multiple analyses with increasing numbers of plate elements
2. Compare the results between each successive run
3. If the results in the **Results | Advanced** tab are changing significantly, your results have not converged and may not be accurate.
4. Repeat the mesh refinement until your results stabilize.
5. If results do not converge even with a very large number of plates, consider ways to simplify or improve your model.

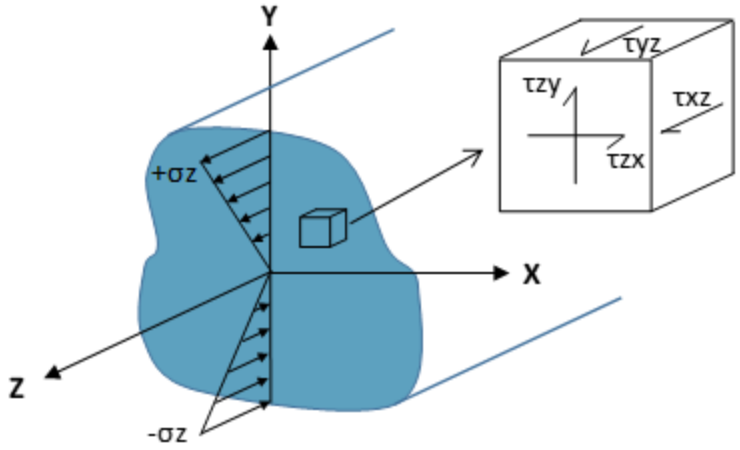
View Graphical Results

You may select among the following for graphical results in the **Project Manager | Analysis Settings** tab:

Result View	Description
Flattened Model	Shows how ShapeBuilder merged touching parts, removed overlaps, and subtracted holes. The model shown is used for analysis.
Mesh	Displays the finite element mesh used to run the analysis. You might look at this to judge element sizes, to find meshing problems or issues (you may zoom and pan the view).
Normal Stress, σ_z	Total normal (Z-direction) stress, from combined axial and bending.
St. Venant, τ_{xz}	X-direction St. Venant shear stress as a result of applied or incidental torsion.
St. Venant, τ_{yz}	Y-direction St. Venant shear stress as a result of applied or incidental torsion
Warping Function	Warping normal function
Flexural, τ_{xz}	X-direction flexural shear stress from applied shear.
Flexural, τ_{yz}	Y-direction flexural shear stress from applied shear.
Combined, τ_{xz}	X-direction total shear stress found by combining flexural and torsional shear stresses. Sign of the stress is used in the combination and the sign convention is shown in the figure below.
Combined, τ_{yz}	Y-direction total shear stress found by combining flexural and torsional shear stresses. Sign of the stress is used in the combination and the sign convention is shown in the figure below.
Resultant, τ	Total Resultant Shear Stress found by taking the square root of the square of the total stresses. It is a positive number. The resultant represents the vector resultant value.

Stress Sign Convention

The orientation and sign convention of the resulting stresses is shown below. All stresses shown are positive; note that a normal stress in tension is designated as positive. ShapeBuilder reports only the shear stresses on the Z face of the cube below.



3.3 Simple Results

Simple section properties are calculated for the overall shape and displayed in the **Results | Simple** tab. When the shape is composed of multiple parts, some of the individual part properties are also shown in the **Results | Simple** tab. Selecting one part causes more of that part's properties to appear, in the **Project Manager | Modify** tab. For shapes that include multiple materials, the "base" material can be specified. The properties for the composite shape are then transformed with a modular ratio. A single asterisk (*) indicates that transformed properties use a modular ratio of *n = E/E_base*. A double asterisk (**) indicates that transformed properties use a modular ratio of *n = Fy/Fy_base* (when a material has multiple values for Fy, the minimum value is used in the modular ratio). For example, areas of each part are weighted by the modular ratio of *n = E/E* base as shown in the equation below.

$$A^* = \sum_{i=1}^{n} \frac{E_i}{E_{base}} A_i$$

Overall

Depth	Overall maximum dimension in the Y-direction.
Width	Overall maximum dimension in the X-direction.
Perimeter	The distance around the boundary of the shape. If holes overlap the outer perimeter to form a notch, it is the "flattened" perimeter that is calculated. Switch to the Analysis View to see the flattened shape.
Weight	Weight per unit-length of the net section, if it were extruded out of plane.
E_{base}	Base modulus of elasticity. Used in calculations having transformed properties (*n = E/E_base*).
F_{y_base}	Base yield stress. Used in calculations having transformed properties (*n = Fy/F_base*)

Geometric Properties

Area

Gross or full cross-sectional area of the shape, less the area of any holes.

Moments of Inertia, I_x, I_y

Also know as the second moment of the area. A measure of the stiffness of the cross-section and its ability to resist bending moments.

$$I_x = \int_A y^2 dA \qquad I_y = \int_A x^2 dA$$

Product of Inertia, I_{xy}

The product of inertia is zero when the x-y axes are the principle axis and is defined by the following equation:

$$I_{xy} = \int_A xy dA$$

Radius of Gyration, r_x, r_y

Radius of gyration is the distance from a reference axis to a point at which the entire area may be concentrated and still have the same moment of inertia as the distributed area. It is used as a measure of the stability of a column.

$$r_x = \sqrt{\frac{I_x}{A}} \qquad r_y = \sqrt{\frac{I_y}{A}}$$

Section Modulus, S_{x+}, S_{x-}, S_{y+}, S_{y-}

The section modulus is useful for calculating the extreme bending stress where I is the moment of inertia about the axis in question and c is the distance from the centroid to the extreme fiber. The plus (+) and minus (-) signs indicate the direction of c from the centroid. For example, Sx+ is the section modulus about the x-axis in the positive y direction.

$$S_x = \frac{I_x}{c} \qquad S_y = \frac{I_y}{c}$$

Center of Gravity, X_c, Y_c

Also known as the Centroid of Area. The point where the moment of the area is zero about any axis as defined below, where the terms $\bar{x}$ and $\bar{y}$ represent the moment arms for the centroid of the differential element that is used. This point is measured from the global XY axes and is labeled CG in the Sketch View.

$$X_c = \frac{\int_A \bar{x} dA}{\int_A dA} \qquad Y_c = \frac{\int_A \bar{y} dA}{\int_A dA}$$

Principal Properties

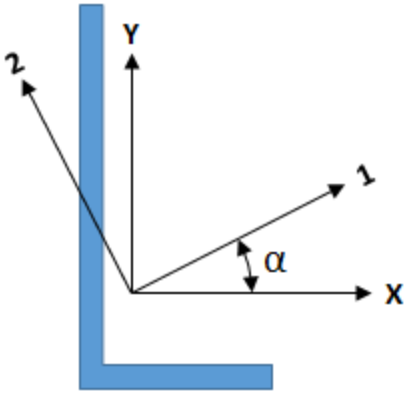
Principal Axes Angle, α

The axes orientation at which the maximum and minimum moments of inertia are obtained, positive according to the right-hand rule. This is the system on which all flexure formulas are based. Moment of inertia Ixy is zero with respect to this coordinate system. Alpha is the angle of

rotation from ShapeBuilder's geometric (X-Y) axes to the principal axes (see the figure below).

Major & Minor Moments of Inertia, I_1, I_2

The maximum (I_1) and minimum (I_2) moments of inertia defined on the principal coordinate system axes (see the figure below). The product of inertia with respect to these axes is always zero.



Polar Properties

Polar Moment of Inertia, I_p

Moment of inertia with respect to the z-axis (normal to the section plane). This is equivalent to the torsion constant, J, **for circular cross-sections only**. In other cases, the polar moment of inertia is larger (perhaps much larger) than the torsion constant.

$$I_p = I_x + I_y$$

Polar Radius of Gyration, r_p

The polar radius of gyration about the centroid of the shape.

$$r_p = \sqrt{\frac{I_p}{A}}$$

Plastic Properties

Plastic Modulus, Z_x, Z_y

The plastic section modulus is the arithmetical sum of the statical moments about the plastic neutral axis of the parts of the section above and below that axis. This term is used to calculate the plastic moment capacity of a section *Mpx* = σy**Zx,* where σ*y* is the yield stress of the material. When a section is composed of multiple materials of differing yield stresses, the *Zx* and *Zy* values are calculated by weighting each part with its yield stress referenced to *Fy_base*.

$$Z_x = A_{comp}\bar{y}_{comp} + A_{tens}\bar{y}_{tens} \qquad Z_y = A_{comp}\bar{x}_{comp} + A_{tens}\bar{x}_{tens}$$

Plastic Neutral Axis, X_{pna}, Y_{pna}	The plastic neutral axis is located such that the equation below is true. If the section is composed of multiple materials of differing yield stresses, the *Fy_comp* and *Fy_tens* values may vary across the cross-section. If the section is composed of only one material, the PNA divides the area of the section into equal halves and is not always located at the centroid. $$A_{comp}F_{y\_comp} = A_{tens}F_{y\_tens}$$

Part Properties

Area	Gross or full cross-sectional area of the part, less the area of any holes.
Major & Minor Moments of Inertia, I_1, I_2	The maximum (*I1*) and minimum (*I2*) moment of inertia of the part about its principal axis (see the figure above).
Center of Gravity, X_c, Y_c	The distance from the part's centroid to the global origin. $$X_c = \frac{\int_A \bar{x}dA}{\int_A dA} \qquad Y_c = \frac{\int_A \bar{y}dA}{\int_A dA}$$

3.4 Advanced Results

FEA Mesh

Nodes	The number of nodes used in the analysis mesh.
Elements	The number of elements used in the analysis mesh.
Largest Element	Area of the largest plate element in the mesh, determines mesh refinement

Normal Stresses

Normal stress is calculated for composite sections with multiple materials.

Combined Axial & Flexural Stress, σ_z	Normal stress (combined axial and bending). Normal stress is positive for tension, negative for compression. $$\sigma_z = \sigma_a + \sigma_b$$ where: $$\sigma_a = \frac{P}{A} \quad \text{and} \quad \sigma_b = \left(\frac{M_x I_y + M_y I_{xy}}{I_x I_y - I_{xy}^2}\right) y - \left(\frac{M_y I_x + M_x I_{xy}}{I_x I_y - I_{xy}^2}\right) x$$

Torsion Properties

The accuracy of these calculated values depends upon the mesh density used in the finite element analysis, see Analysis

Settings for more details on improving the accuracy of your results. Advanced analysis is not performed on a shape composed of multiple disconnected parts or for composite shapes (i.e when multiple materials are defined). The theory is based on a single material and the warping normal functions. Beware of using "superposition" of your parts independently to estimate a *J* value, as your results may not be correct.

Polar Radius of Gyration about the Shear Center, r_o

The polar radius of gyration about the shear center is defined in AISC 360 Specification Chapter E Commentary as:

$$r_o = \sqrt{\frac{I_x + I_y + A(X_{sc}^2 + Y_{sc}^2)}{A}}$$

Note: This property is only shown if the shear center does not coincide with the centroid. See also r_p in the Simple Results.

AISC Flexural Constant, *H*

Derived from the polar radius of gyration about the shear center is the AISC flexural constant:

$$H = 1 - \frac{(X_{sc}^2 + Y_{sc}^2)}{r_o^2}$$

Refer to the AISC manual for more details. *H*=1 if the shear center and centroid coincide.

Shear Center, X_{sc}, Y_{sc}

Also known as the Flexural Center. The Shear Center is the point on the cross section where an applied shear force will cause no twisting of the cross section as it bends. In general, this is not the centroid. If the section is symmetric, the shear center will lie on the axis of symmetry; for doubly symmetric sections, the shear center will coincide with the centroid. This point is located with respect to the global origin.

Warping Constant, C_w

Warping constant is calculated as:

$$C_w = \int_A \omega^2 dA$$

where ω is the warping function

Torsion Constant, *J*

Torsional stiffness factor is a more accurate measure of the torsional rigidity than polar moment of inertia. The approximate equations for thin-walled open sections and thin-walled closed sections are given below. ShapeBuilder uses a sophisticated numerical process to calculate the value, which may be quite different than the approximation formula results.

$$J = \sum \left(\frac{b \cdot t^3}{3} \right) \text{ (open section)} \qquad J = \frac{4A_o^2}{\int_0^{L_o} \frac{ds}{t}} \text{ (closed section)}$$

Monosymmetry Factor, B_1

Used for calculating lateral torsional buckling of singly-symmetric cross sections. [See reference Trahair and Nethercot].

$$B_1 = \frac{1}{I_x}\int_A y(x^2 + y^2)dA - 2y_o$$

More information on the above parameters can be found in the References section.

Shear Stress Results (Minimum & Maximum)

These require the full FEA analysis.

St. Venant, τ_{xy}, τ_{yz}	Shear stress due to torque only. These stresses are also called uniform torsional shear stresses.
Warping Function	The normalized warping function (length^2). This is not a stress and it is not load-dependent.
Flexural, τ_{xy},τ_{yz}	Shear stresses induced by the applied shear loads.
Combined, τ_{xy}, τ_{yz}	The superposition of Flexural and St.Venant shear stresses.
Resultant, τ	Resultant shear stress found by taking the square root of the sum of the squares of the Flexural and St. Venant shear stresses. The resultant represents the vector resultant value that is a positive number.

Shear Flow

First Moment of Area, *Qx, Qy*	The first moment of area of the individual part or the region defined by the shear flow location about the centroid of the entire built-up shape. The first moment of area is used to determine the shear flow (*VQ/I*). $Q_x = \int_A y dA \quad Q_y = \int_A x dA$
Shear Flow, $f(V_x)$, $f(V_y)$	The shear flow due to the shear force in the X-direction (*Vx*) and the shear force in the Y-direction (*Vy*). These values are calculated for each part and for the regions defined by the shear flow locations. $f(V_x) = \frac{V_x Q_y}{I_y} \quad f(V_y) = \frac{V_y Q_x}{I_x}$

4 Script

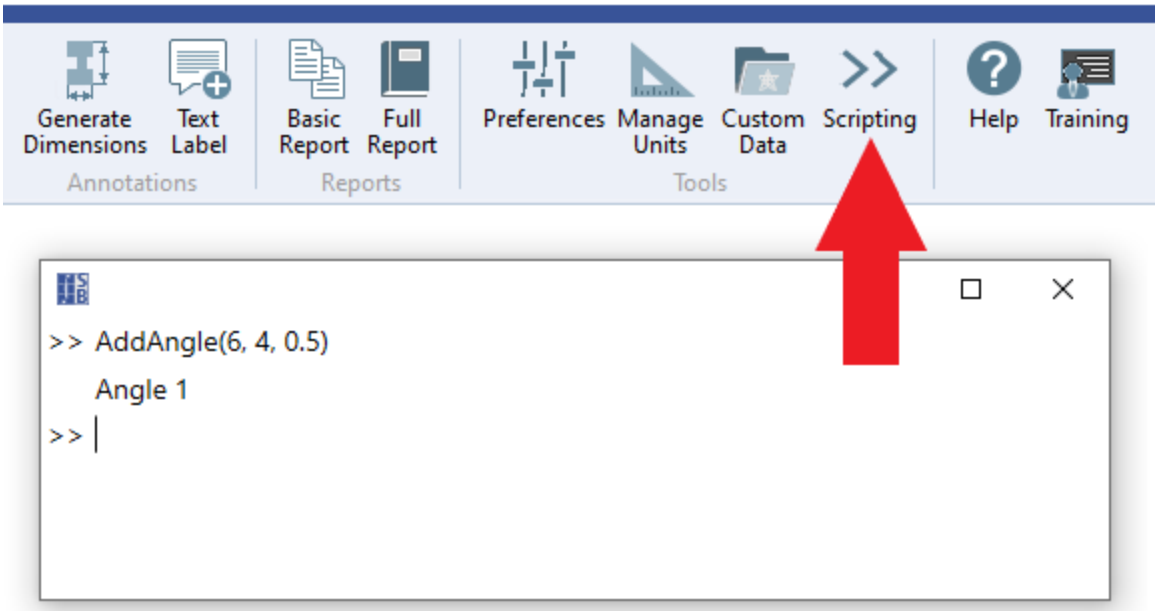
4.1 Script Overview

Introduction

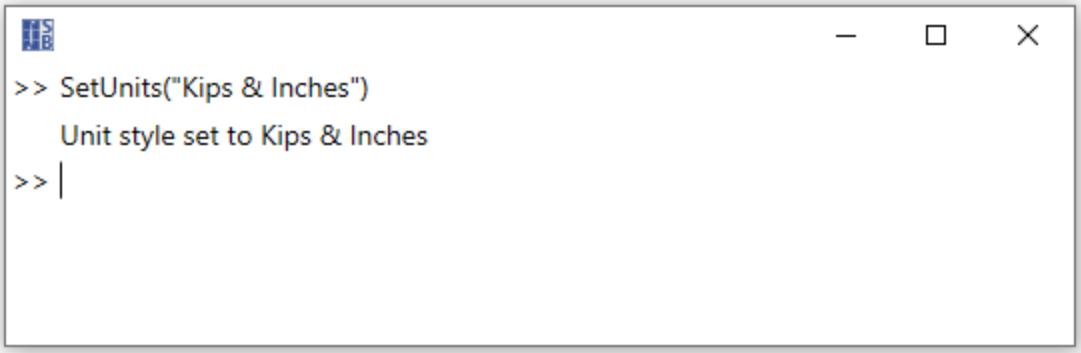
The script feature in ShapeBuilder is a powerful tool used to add and modify shapes, apply loads, extract results, generate reports etc. using a command line interface. In addition to supporting the various Commands outlined in this Help File, the command line will accept any valid command in the C# Programming language (allowing the use of if statements, for loops, etc.). In addition to using the command line directly, more complex Scripts can be generated in any text file and read into the program. This allows shapes with complex geometry to be created, parametric studies to be performed, finite element meshes to be automatically refined until convergence, batch processing of CAD files, etc.

Command Line Basics

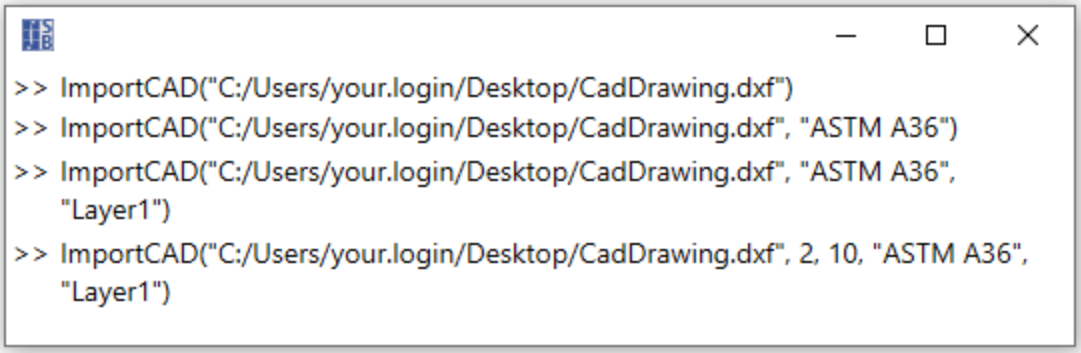
1. Location
    a. The Script tool is accessed by clicking the Scripting button in the Home ribbon.



2. Units
    a. Input and output are always in the current unit style. The style can be changed from the command line.



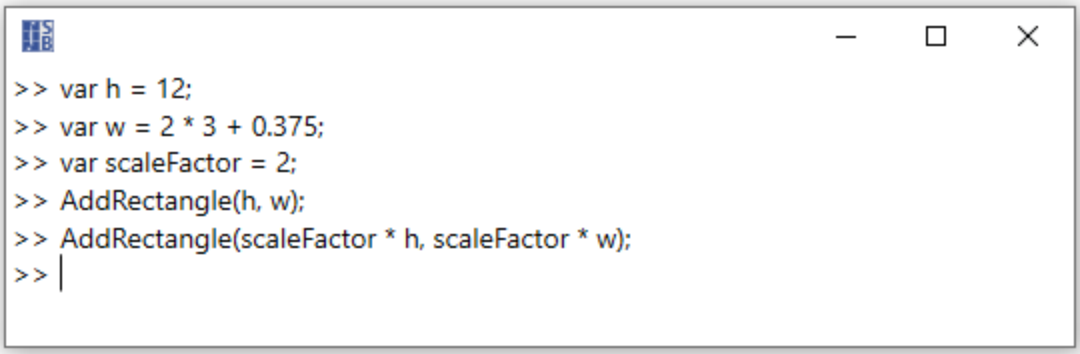
3. Import CAD
    a. There are various options available when importing CAD files from the command line (see the Commands page for a complete list of the available parameters that the ImportCAD() command can accept). Note: Paths must be enclosed in quotes and must contain forward slashes (/) not backslashes (\).



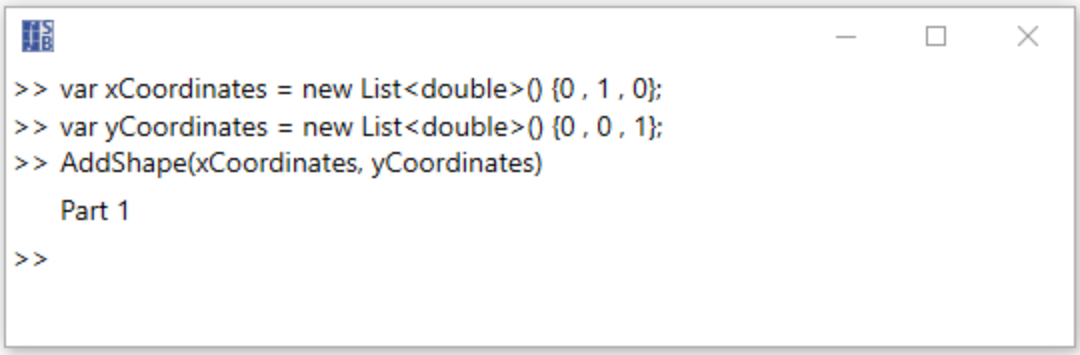
4. Add Shapes
    a. Add any of the simple, rolled, and bent parametric shapes to the model from the command line (see the Commands page for a complete list of the available parametric shapes). The PickDatabaseSection() command can be used to launch the Shape Database dialog box to choose a shape from the database.

    b. The return value can be suppressed by using a semi-colon on at the end of the command.

c. Information can be stored in variables and math can be performed in the command line. Variables circumvent the need to input the same number repeatedly. The example below uses variables and math in the command line to define the height and width of the first rectangle which is then scaled to create the second rectangle. Note: When storing a value in a variable, the line must end with a semicolon.



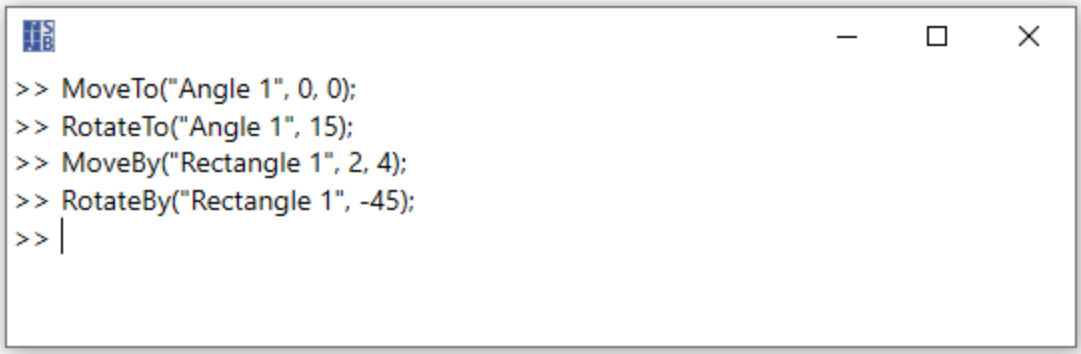
d. A shape can be added to the model from lists of the X and Y coordinates.



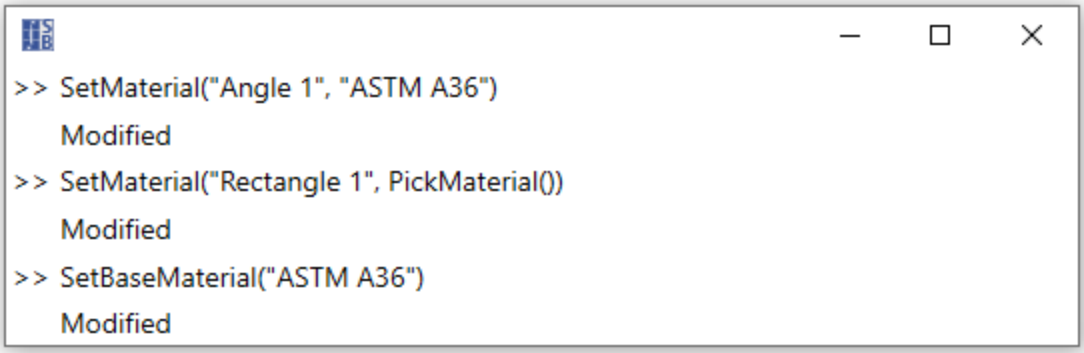
5. Modify Shapes
   a. Selection: Specific shapes can be selected using the Select() call. Note: Shapes and other item names must be enclosed in quotes. The SelectAllShapes() call selects every shape in the model.

b. Delete: The selected shapes can be removed using the DeleteSlected() command or specific shapes can be deleted by name.

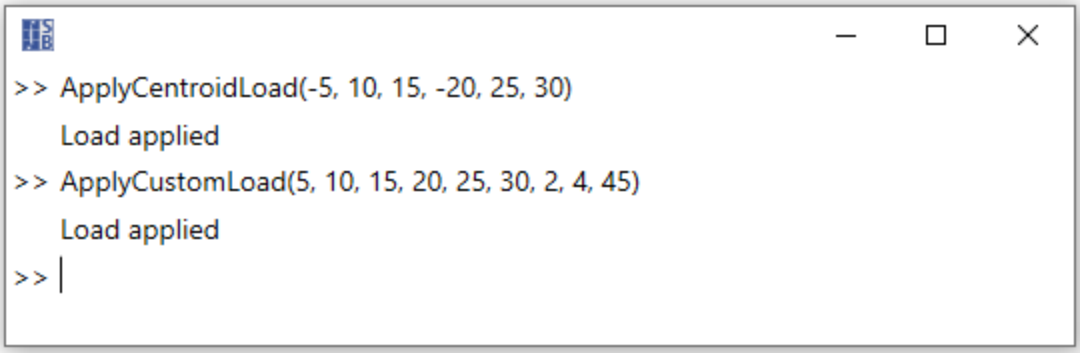
c. Move & Rotate: Shapes can be moved and rotated to a certain value or by a certain value using the command line.



d. Set Material: The material can be set for a specific shape and the base material can be defined for the model. The PickMaterial() command can be used to launch the Material Database to choose a material if its exact name is unknown.



6. Apply Loads & Mesh Refinement
    a. Normal force, shear forces, moments, and torsion can be applied to the cross-section using coordinate systems (see the Commands page for a complete list).

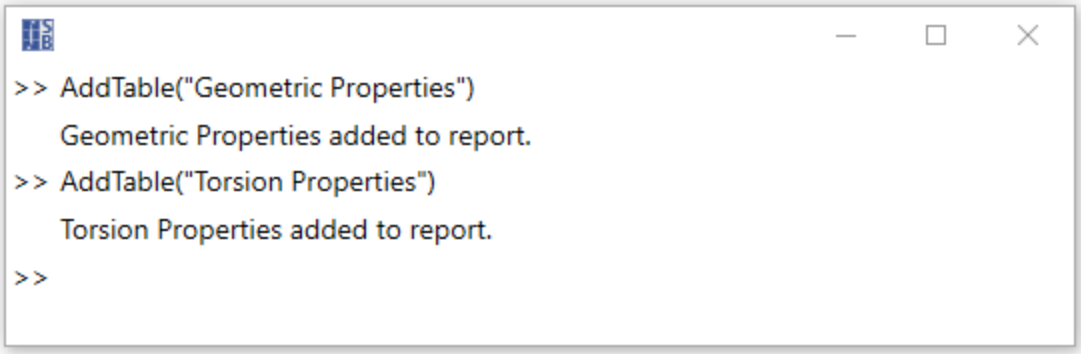


    b. The finite element mesh size can be set from the command line. Note: The mesh refinement is exponential where 0 = coarse and 10 = fine..

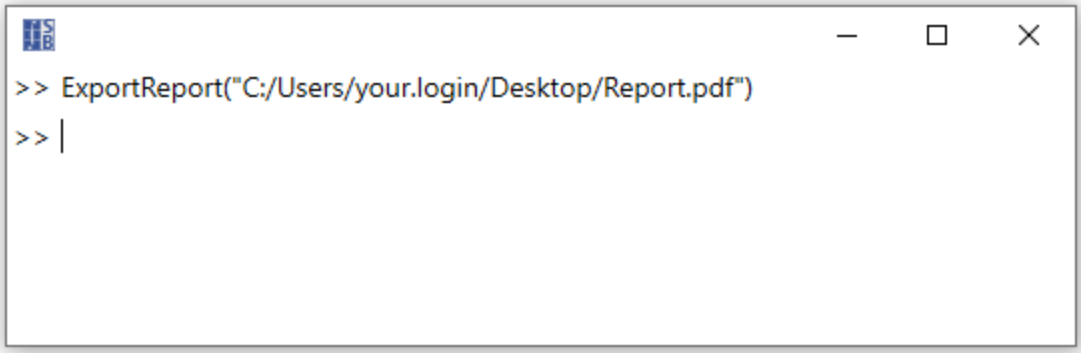
7. Obtain Results
    a. The various section properties can be obtained directly using the command line.

b. The maximum or minimum values for the various stresses and the warping function can obtained directly using the command line.

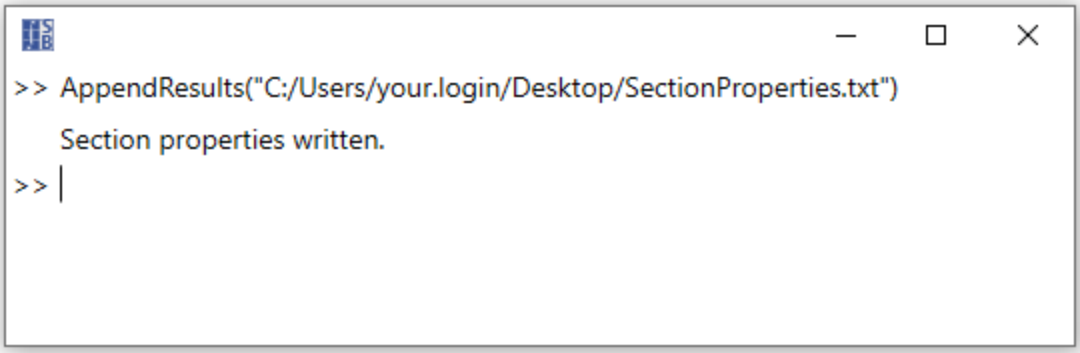
8. Reports
   a. Add Tables: The various tables can be added to the report via the command line.



   b. Export Reports: Reports can be exported to a specified location.



c. Append Results: The section properties can be added to text file at a specified path.

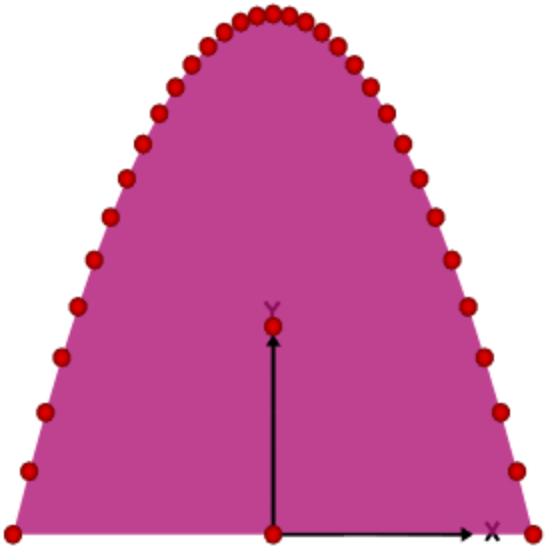


9. Miscellaneous
   a. Integer Math: In the command line, the quotient of two integers is an integer which may not produce the intended result. See the C# Numeric Types Tutorial for more information.

C# Programming

The command line accepts any valid command in the C# Programming Language, allowing the use of if statements, for loops, etc. In the example below, a command is defined to describe the y coordinate of a parabola terms of x. The Pow(Double, Double) method of the .Net System Math Class is used to perform squared operation. A four loop is used to add the x and y coordinates of the shape to lists which are then used to define the shape. In addition to using the command line to program directly, more complex Scripts can be generated in any text file and read into the program.

```
>> double y(double x) => 16 - Math.Pow(x, 2)/4.0;
>> var xCoordinates = new List<double>();
>> var yCoordinates = new List<double>();
>> for(double x = -8; x <= 8; x = x + 0.5) {xCoordinates.Add(x);
   yCoordinates.Add(y(x));}
>> AddShape(xCoordinates, yCoordinates)
   Part 1
```



4.2 Commands

Script Command Categories

- User Interface
- General
- Import CAD
- Add Shapes
- Modify
- Loads
- Analysis
- Results
- Report

	Command	Description	Example Input	Example Result
User Interface (BACK TO TOP)	Clear	Clears all text in the command line and clears any stored variables		
	Browse	Launches the Open dialog box to navigate to an external script to run		
	Up/Down Arrows	Use the "*Up Arrow*" and "*Down Arrow*"		

		keys to navigate the command line history		
	Esc	Press the "*Esc*" key while a script is running to end the script run		
General (BACK TO TOP)	Help()	Launches the Help File and navigates to the script overview page		
	Help(command)	Launches the Help File and navigates to the specified command	Help("AddDatabaseShape")	Launches the Help File and navigates to the AddDatabaseShape command
	SetUnits(style)	Sets the unit style to a default or custom style in the program	SetUnits("Canadian")	Sets the programs unit style to Canadian
	GenerateDimensions()	Creates typical dimensions for overall height and width and distances to the centroid		
	ZoomExtents()	Zooms to fit the entire cross-section in the window		
	Print(List<names>)	Prints the list of items in a comma-delimited format	Print(Shapes())	Prints the list of shapes in the project in a comma-delimited format
	List(List<names>)	Lists the list of items with one item per line	List(Shapes())	Lists the list of shapes with one item per line
	SaveAs(path)	Saves the ShapeBuilder file to a specified path	SaveAs("C:/Users/your.login/Desktop/ShapeFile.sbp")	Saves the current ShapeBuilder file to the desktop as ShapeFile.sbp
	PickMaterial()	Opens the Material Database dialog box	SetMaterial("S1", PickMaterial())	Opens the Material Database dialog box to define a material for shape S1
	PickDatabaseSection()	Opens the Shape Database dialog box	AddDatabaseShape(PickDatabaseSection())	Opens the Shape Database dialog box to define a database section to be added to the model
Import CAD (BACK TO TOP)	GetCADFiles(folder)	Returns a list of CAD files in a specified folder	List(GetCADFiles("C:/Users/your.login/Desktop/CADFolder"))	Prints a list of the cad files located in CADFolder on the desktop
	ImportCAD(path)	Imports the CAD file located at the specified path	ImportCAD("C:/Users/your.login/Desktop/CadDrawing.dxf")	Imports the CadDrawing.dxf file from the desktop
	ImportCAD(path, material)	Imports the CAD file located at the specified path with the specified material	ImportCAD("C:/Users/your.login/Desktop/CadDrawing.dxf", "ASTM A36")	Imports the CadDrawing.dxf file from the desktop and sets the material to ASTM A36
	ImportCAD(path, material, layer)	Imports the specified layers	ImportCAD("C:/Users/your.login/Desktop/CadDrawing.dxf", "ASTM A36", "Layer1")	Imports Layer1 from the CadDrawing.dxf

		from the CAD file located at the specified path with the specified material		file from the desktop and sets the material to ASTM A36
	ImportCAD(path, multiplier, arcDivisions, material, layer)	Imports the specified layers from the CAD file located at the specified path with the specified material	ImportCAD("C:/Users/your.login/Desktop/CadDrawing.dxf", 2, 10, "ASTM A36", "Layer1")	Imports Layer1 (scaled by a factor of two) from the CadDrawing.dxf file from the desktop and sets the material to ASTM A36. Ten line segments will be used to approximate arc and circle shapes.
Add Shapes (BACK TO TOP)	Shapes()	Returns a list of all the shapes in the project	Print(Shapes())	Prints the list of all the shapes in the project in a comma-delimited format
	AddShape(List, List)	Adds a shape to the model based on lists of the X and Y coordinates	AddShape(new List<double>() {0 , 1 , 0}, new List<double>() {0 , 0 , 1})	Adds a triangle to the model with points at {0, 0}, {1, 0}, and {0, 1}
	AddDatabaseShape(name)	Adds a database shape to the model	AddDatabaseShape("L5x5x1/2")	Adds a L5x5x1/2 to the model
	AddAngle(verticalLeg, horizontalLeg, thickness)	Adds a parametric angle to the model	AddAngle(5, 5, 0.5)	Adds a L5x5x1/2 angle to the model
	AddChannel(depth, width, tf, tw)	Adds a parametric channel to the model	AddChannel(10, 3, 0.5, 0.25)	Adds a 10 in deep by 3 inch wide channel to the model
	AddCircle(diameter)	Adds a parametric circle to the model	AddCircle(6)	Adds a 6 in diameter circle to the model
	AddDoubleAngle(depth, totalWidth, thickness, spacing)	Adds a parametric double angle to the model	AddDoubleAngle(5, 10, 0.5, 0)	Adds a 2L5x5x1/2 double to the model
	AddFillet(radius)	Adds a parametric fillet to the model	AddFillet(0.75)	Adds a 0.75 in radius fillet to the model
	AddHat(depth, width, lip, thickness)	Adds a parametric hat to the model	AddHat(5, 2, 1, 0.25)	Adds a 5 in deep by 2 in wide hat section to the model
	AddI(depth, width, tf, tw)	Adds a parametric I-shape to the model	AddI(10, 5, 0.75, 0.5)	Adds a 10 in deep by 5 in wide hat section to the model
	AddIsoscelesTriangle(depth, width)	Adds a parametric isosceles triangle to the model	AddIsoscelesTriangle(4, 3)	Adds a 4 in deep by 3 in wide isosceles triangle to the model
	AddPipe(outsideDiameter, insideDiameter)	Adds a parametric pipe to the model	AddPipe(12, 11.5)	Adds a 12 in outside diameter by 11.5 in inside diameter pipe to the model
	AddPipeSection(radius, thickness, alpha)	Adds a parametric pipe section to the model	AddPipeSection(6, 0.25, 90)	Adds a 90 degree pipe sector with a radius of 6 in and thickness of 0.25 in to the model
	AddQuarterCircle(radius)	Adds a parametric quarter circle to the model	AddQuarterCircle(3)	Adds a quarter circle with a 3 in radius to the model

AddRectangle(depth, width)	Adds a parametric rectangle to the model	AddRectangle(6, 3)	Adds a 6 in deep by 3 in wide rectangle to the model
AddRectangularTube(depth, width, thickness)	Adds a parametric rectangular tube to the model	AddRectangularTube(6, 3, 0.25)	Adds a 6 in deep by 3 in wide rectangular tube of thickness 0.25 in to the model
AddRegularPolygon(numberSides, sideLength)	Adds a parametric regular polygon to the model	AddRegularPolygon(8, 2)	Adds an octagon with 2 in sides to the model
AddRightTriangle(depth, width)	Adds a parametric right triangle to the model	AddRightTriangle(4, 2)	Adds a 4 in deep by 2 in wide right triangle to the model
AddSpandrel(depth, width, stemThickness, flangeThickness)	Adds a parametric spandrel to the model	AddSpandrel(12, 24, 18, 4)	Adds a 12 in deep by 24 in wide spandrel to the model
AddTaperedTee(depth, width, tf, twt, twb)	Adds a parametric tapered tee to the model	AddTaperedTee(8, 6, 0.75, 0.75, 0.5)	Adds a 8 in deep by 6 in wide tapered tee to the model
AddTee(depth, width, tf, tw)	Adds a parametric tee to the model	AddTee(8, 6, 0.75, 0.5)	Adds a 8 in deep by 6 in wide tee to the model
AddTrapezoid(depth, baseWidth, leftOffset, rightOffset)	Adds a parametric trapezoid to the model	AddTrapezoid(6, 4, 2, 1)	Adds a 6 in deep by 4 in wide trapezoid to the model
AddZee(depth, width, tf, tw)	Adds a parametric zee to the model	AddZee(6, 3, 0.75, 0.5)	Adds a 6 in deep by 3 in wide zee to the model
AddRolledAngle(depth, width, thickness, centerFillet, edgeFillet)	Adds a parametric rolled angle to the model	AddRolledAngle(6, 4, 0.375, 0.5, 0.25)	Adds a 6 in deep by 4 in wide rolled angle to the model
AddRolledChannel(depth, width, tw, tf1, tf2, centerFillet, edgeFillet)	Adds a parametric rolled channel to the model	AddRolledChannel(16, 4, 0.75, 0.70, 0.5, 0.65, 0.375)	Adds a 16 in deep by 4 in wide rolled channel to the model
AddRolledDoubleAngle(depth, width, thickness, centerFillet, edgeFillet, spacing)	Adds a parametric rolled double angle to the model	AddRolledDoubleAngle(6, 12, 0.375, 0.5, 0.25, 0.75)	Adds a 6 in deep by 12 inch wide rolled double angle to the model
AddRolledI(depth, width, web, flangeCenter, flangeEdge, centerFilletRadius, edgeFilletRadius)	Adds a parametric rolled I-shape to the model	AddRolledI(12, 6, 0.25, 0.75, 0.625, 0.5, 0.25)	Adds a 12 in deep by 6 in wide rolled I-shape to the model
AddRolledRectangle(depth, width, radius)	Adds a parametric rolled rectangle to the model	AddRolledRectangle(10, 5, 1)	Adds a 10 in deep by 5 in wide rolled rectangle to the model
AddRolledTee(depth, width, topWebThickness, bottomWebThickness, flangeCenterThickness, flangeEdgeThickness, centerFillet, flangeFillet, webFillet)	Adds a parametric rolled tee to the model	AddRolledTee(8, 6, 0.375, 0.25, 0.625, 0.5, 0.25, 0.125, 0.0625)	Adds a 8 in deep by 6 in wide rolled tee to the model
AddRolledTube(depth, width, thickness, insideFillet, outsideFillet)	Adds a parametric rolled tube to the model	AddRolledTube(8, 4, 0.25, 0.25, 0.5)	Adds a 8 in deep by 4 in wide rolled tube to the model
AddRolledZee(depth, width, tf, tw, edgeFillet, centerFillet)	Adds a parametric rolled zee to the model	AddRolledZee(6, 4, 0.375, 0.25, 0.125, 0.375)	Adds a 6 in deep by 4 in wide rolled zee to the model

	AddBentAngle(verticalLeg, horizontalLeg, thickness, insideRadius)	Adds a parametric bent angle to the model	AddBentAngle(6, 4, 0.25, 0.125)	Adds a 6 in deep by 4 in wide bent angle to the model
	AddBentAngleLips(verticalLeg, horizontalLeg, thickness, insideRadius, lip)	Adds a parametric bent angle w/ lips to the model	AddBentAngleLips(6, 4, 0.25, 0.125, 1)	Adds a 6 in deep by 4 in wide bent angle with 1 in lips to the model
	AddBentChannel(depth, width, thickness, insideRadius)	Adds a parametric bent channel to the model	AddBentChannel(8, 4, 0.25, 0.25)	Adds a 8 in deep by 4 in wide bent channel to the model
	AddBentChannelLips(depth, width, thickness, insideRadius, lip)	Adds a parametric bent channel w/ lips to the model	AddBentChannelLips(8, 4, 0.25, 0.25, 0.75)	Adds a 8 in deep by 4 in wide bent channel with 0.75 in long lips to the model
	AddBentEaveStrut(depth, topWidth, bottomWidth, thickness, insideRadius, lip, topFlangeAngle, bottomFlangeAngle, isLipParallelToWeb)	Adds a parametric bent eave strut to the model	AddBentEaveStrut(8, 4, 4, 0.25, 0.375, 0.5, 75, 90, true)	Adds a 8 in deep by 4 in wide bent eave strut to the model
	AddBentHat(depth, width, opening, thickness, radius)	Adds a parametric bent hat to the model	AddBentHat(6, 2, 3, 0.25, 0.25)	Adds a 6 in deep by 2 in wide bent hat to the model
	AddBentZee(depth, width, thickness, radius)	Adds a parametric bent zee to the model	AddBentZee(8, 4, 0.25, 0.25)	Adds a 8 in deep by 4 in wide bent zee to the model
	AddBentZeeLips(depth, width, thickness, radius, lip, lipAngle)	Adds a parametric bent zee w/ lips to the model	AddBentZeeLips(8, 6, 0.25, 0.25, 0.5, 15)	Adds a 8 in deep by 6 in wide bent zee with 0.5 in lips to the model
Modify (BACK TO TOP)	Select(names[])	Selects a specified item(s) in the model	Select("S1", "S2")	Selects shapes S1 and S2 in the model
	SelectAllShapes()	Selects all of the shapes in the model		
	UnselectAll()	Unselects everything in the model		
	DeleteSelected()	Deletes the selected shape(s) in the model	Select shapes S1 and S2 then enter DeleteSelected()	Deletes shapes S1 and S2 in the model
	Delete(names[])	Deletes a specified shape(s) in the model	Delete("S1", "S2")	Deletes shapes S1 and S2 in the model
	ZeroCentroid()	Locates the cross-section's center-of-gravity (cg) at the global origin		
	MovePointTo(X, Y)	Moves a point that defines a generic section part to modify the shape's boundary	Select a point of a generic section then enter MovePointTo(5, 10)	Moves the selected point to (5, 10)
	MoveBy(shape, distanceX, distanceY)	Moves a specified shape by a specified distance in each global direction	MoveBy("S1", 5, 10)	Moves shape S1 by 5 inches and 10 inches in the global X and Y-directions, respectively
	MoveTo(shape, X, Y)	Moves a specified shape to a defined location	MoveTo("S1", 0, 0)	Moves shape S1 to the origin

	RotateBy(shape, distanceX, distanceY)	Rotates a specified shape by a specified amount about its centroid (CCW is +)	RotateBy("S1", 15)	Rotates shape S1 by 15 degrees CCW
	RotateTo(shape, X, Y)	Rotates a specified shape to a specified angle amount about its centroid (CCW is +)	RotateTo("S1", -15)	Rotates shape S1 to 15 degrees CW
	RotateToHorizontal()	Rotates selected (or all) shapes to align two selected points with the X axis		
	RotateToVertical()	Rotates selected (or all) shapes to align two selected points with the Y axis		
	SetMaterial(shape, material)	Sets the material for a specified shape in the model	SetMaterial("S1", "ASTM A36")	Sets the material for shape S1 to ASTM A36 steel
	SetBaseMaterial(material)	Sets the base material in the model	SetBaseMaterial("ASTM A992 Grade 50")	Sets the base material in the model to ASTM A992 Grade 50 steel
	ToggleHole(shape)	Toggles the hole parameter for a specified shape in the model	ToggleHole("S1")	Sets shape S1 as a hole in the model
Loads (BACK TO TOP)	ApplyCentroidLoad(p, va, vb, ma, mb, t)	Applies specified load(s) to the shape's centroid where the primary axis is parallel to the global X-axis and the secondary axis is parallel to the global Y-axis	ApplyCentroidLoad(1, 2, 3, 4, 5, 6)	Applies 1 kip tension, 2 kips shear along primary axis, 3 kips shear along secondary axis, 4 kips-in moment about primary axis, 5 kip-in moment about secondary axis, and 6 kip-in torsional moment to the section (assuming the Unit Style is Kip & Inches).
	ApplyShearCenterLoad(p, va, vb, ma, mb, t)	Applies specified load(s) to the shape's shear center where the primary axis is parallel to the global X-axis and the secondary axis is parallel to the global Y-axis	ApplyShearCenterLoad(-1, 2, 3, 4, 5, 6)	Applies 1 kip compression, 2 kips shear along primary axis, 3 kips shear along secondary axis, 4 kips-in moment about primary axis, 5 kip-in moment about secondary axis, and 6 kip-in torsional moment to the section (assuming Unit Styles are Kip & Inches)
	ApplyPrincipalLoad(p, va, vb, ma, mb, t)	Applies specified load(s) to the shape's centroid where the primary axis is the shape's major principal axis	ApplyPrincipalLoad(1, 2, 3, 4, 5, 6)	Applies 1 kip tension, 2 kips shear along primary axis, 3 kips shear along secondary axis, 4 kips-in moment

		and the secondary axis is the shape's minor principal axis		about primary axis, 5 kip-in moment about secondary axis, and 6 kip-in torsional moment to the section (assuming the Unit Style is Kip & Inches).
	ApplyGlobalLoad(p, va, vb, ma, mb, t)	Applies specified load(s) at the global origin where the primary axis is the global X-axis and the secondary axis is the global Y-axis	ApplyGlobalLoad(1, 2, 3, 4, 5, 6)	Applies 1 kip tension, 2 kips shear along primary axis, 3 kips shear along secondary axis, 4 kips-in moment about primary axis, 5 kip-in moment about secondary axis, and 6 kip-in torsional moment to the section (assuming Unit Styles are Kip & Inches)
	ApplyCustomLoad(p, va, vb, ma, mb, t, x, y, angle)	Applies specified load(s) to a specified location with the primary axis and secondary axis rotated a specified amount	ApplyCustomLoad(1, 2, 3, 4, 5, 6, 10, 20, 45)	Applies 1 kip tension, 2 kips shear along primary axis, 3 kips shear along secondary axis, 4 kips-in moment about primary axis, 5 kip-in moment about secondary axis, and 6 kip-in torsional moment to the section at {10 in, 20 in} where the primary and secondary axes are rotated 45 degrees ccw (assuming Unit Styles are Kip & Inches & Degrees)
Analysis (BACK TO TOP)	SimpleAnalysis()	Pauses the script until the simple analysis completes. Note: Only used for external scripts and must be preceded by "await"	await SimpleAnalysis();	Pauses the external script until the simple analysis completes
	AdvancedAnalysis()	Pauses the script until the advanced analysis completes. Note: Only used for external scripts and must be preceded by "await"	await AdvancedAnalysis();	Pauses the external script until the advanced analysis completes
	SetMesh(ratio)	Sets the finite element mesh size. Note: The mesh refinement is exponential where 0 = coarse and 10 = fine.	SetMesh(0)	Sets the mesh refinement to coarse
Results	GetSectionProperty(property)	Returns a specified	GetSectionProperty("Ix")	Returns the moment

(BACK TO TOP)		section property		of inertia bout the centroidal X-axis
	GetPartProperty(part, property)	Returns a specified section property for a specified part	GetSectionProperty("Angle", "Area")	Returns the area for the angle section part
	Nodes()	Returns the number of meshed nodes in the model		
	Elements()	Returns the number of meshed elements for the shape		
	NormalStress(isMax)	Returns the shape's maximum or minimum normal stress	NormalStress(false)	Returns the shape's minimum normal stress
	StVenantXZ(isMax)	Returns the shape's maximum or minimum St. Venant τxz stress	StVenantXZ(true)	Returns the shape's maximum St. Venant τxz stress
	StVenantYZ(isMax)	Returns the shape's maximum or minimum St. Venant τyz stress	StVenantYZ(false)	Returns the shape's minimum St. Venant τyz stress
	FlexuralShearXZ(isMax)	Returns the shape's maximum or minimum flexural τxz shear stress	FlexuralShearXZ(true)	Returns the shape's maximum flexural τxz shear stress
	FlexuralShearYZ(isMax)	Returns the shape's maximum or minimum flexural τyz shear stress	FlexuralShearYZ(false)	Returns the shape's minimum flexural τyz shear stress
	CombinedShearXZ(isMax)	Returns the shape's maximum or minimum combined St. Venant τxz and flexural τxz shear stress	CombinedShearXZ(true)	Returns the shape's maximum combined St. Venant τxz and flexural τxz shear stress
	CombinedShearYZ(isMax)	(isMax)Returns the shape's maximum or minimum combined St. Venant τyz and flexural τyz shear stress	CombinedShearYZ(false)	Returns the shape's minimum combined St. Venant τyz and flexural τyz shear stress
	ResultantShear(isMax)	Returns the shape's maximum or minimum resultant shear stress	ResultantShear(true)	Returns the shape's maximum resultant shear stress
	WarpingFunction(isMax)	Returns the shape's maximum or minimum warping function value	WarpingFunction(true)	Returns the shape's maximum warping function value
Report (BACK TO TOP)	PrintResults()	Prints the calculated property results in the command line		
	ExportResults()	Exports the calculated property results to a tab-delimited text file		
	ExportResults(path)	Exports the	ExportResults("C:/Users/your.login/Desktop/Results.txt")	Exports the

		calculated property results to a tab-delimited text file at a specified path		calculated property results to the Results.txt file on the Desktop
	AppendResults(path)	Appends the results to the text file at a specified path	AppendResults("C:/Users/your.login/Desktop/SectionProperties.txt")	Appends the results to the SectionProperties.txt file on the Desktop
	AddTable(title)	Adds a specified table to the report	AddTable("Geometric Properties")	Adds the Geometric Properties table to the report
	AddBasicReport()	Generates a basic report with available graphics and results		
	ExportReport(path)	Exports the results to a specified path	ExportReport("C:/Users/your.login/Desktop/Report.pdf")	Saves the report as a .pdf on the desktop for the specified user

Note: The examples in the table above assume the unit style is "Kips & Inches".

4.3 External Scripts

Running External Scripts

1. Opening External Scripts
    a. Type Browse into the command line to launch the Open dial box and select the script text file to use.
    b. Directly type the path of the script text file into to the command line and press Enter.
2. Example Scripts
    a. Refine Mesh
    b. Optimize Angle
    c. Process CAD
    d. Buckling Equations

4.4 Refine Mesh

Refine Mesh

```
//Try finer and finer meshes until the desired section property stops changing
var property = "J";
var errorTolerance = 0.01; //stop refinement when the change is less than 1%
var step = 1.0; //increase the mesh ratio by this much each iteration

var ratio = 0.0;
var value = -1.0; //initialize to something that certainly won't be correct
var change = 100.0;
while(change > errorTolerance)
{
    ratio += step;
    SetMesh(ratio);
    await AdvancedAnalysis();
    var newValue = GetSectionProperty(property);
    change = Math.Abs((newValue - value)/value);
    value = newValue;
```

```
}

//Print out the final results
$"{property} converged to {value:e3}. Change = {change:p2}"
```

4.5 Optimize Angle

Optimize Angle

```
//Finds the thinnest 4x4 angle that can handle an 8 kip shear force.
//The force is applied at the centroid so shear stresses are a
//combination of flexural shear and torsional shear.

SetUnits("Kips & Inches");

ApplyCentroidLoad(0, 0, -8.0, 0, 0, 0);

var t = 0.0;
var extreme = 100.0;
var allowable = 0.9 * 0.6 * 36;
while(extreme > allowable)
{
    //remove the previous angle (if any)
    SelectAllShapes();
    DeleteSelected();

    //increment the thickness and add the shape
    t += 1/16.0;
    var angle = AddRolledAngle(4, 4, t, t, 0.8*t);
    SetMaterial(angle, "ASTM A36");
    MoveTo(angle, 0, 0);

    //extract the extreme shear stress (flexure + torsion)
    //need to check the mesh refinement as we go
    var ratio = 0.0;
    var value = 0.0;
    var change = 100.0;
    while(change > 0.02)
    {
        ratio += 1.0;
        SetMesh(ratio);
        await AdvancedAnalysis();
        var max = ResultantShear(true);
        var min = ResultantShear(false);
        var newValue = Math.Max(Math.Abs(max), Math.Abs(min));
        change = Math.Abs((newValue - value)/value);
        value = newValue;
    }
    extreme = value;
}

$"t_required = {t:f3} inches; extreme stress = {extreme:f2} ksi"
```

4.6 Process CAD

Process CAD

```
SetUnits("Kips & Inches");

//define where the CAD files are located
var source = @"C:\Users\your.login\Desktop\ShapeBuilder\ToProcess";
//define where to save the generated ShapeBuilder files, repord pdfs, and section
property table
var results = @"C:\Users\your.login\Desktop\ShapeBuilder\Processed\";
//create the result folder if it doesn't already exist
Directory.CreateDirectory(results);

//get a material (from the dialog) for all the imported shapes,
//if you know the path to the material, you could skip the dialog and just "hard code"
it below
var material = PickMaterial();

int count = 0;
foreach(var file in GetCADFiles(source))
{
    //file path, import scale, arc divisions, material, cad layer (null for all layers)
    ImportCAD(file, 1.0, 10, material, null);

    await SimpleAnalysis();
    ZeroCentroid();

    GenerateDimensions();
    UnselectAll();
    ZoomExtents();

    //the following is only needed if you want advanced (torsion) shape properties
    //otherwise just await the SimpleAnalysis here
    SetMesh(4.0); //result convergence at this setting would need to be checked
    await AdvancedAnalysis();

    //add tables to the report
    AddTable("Sketch View");
    AddTable("Geometric Properties");
    AddTable("Overall Properties");
    AddTable("Principal Properties");
    AddTable("Torsion Properties");

    //save the shapebuilder file, a pdf of the report, and update the section
properties table
    var fileName = Path.GetFileNameWithoutExtension(file);
    SaveAs($"{results}{fileName}{".sbf"}");
    ExportReport($"{results}{fileName}{".pdf"}");
    AppendResults($"{results}{"Table.txt"}");

    //clear the project before importing the next shape
    SelectAllShapes();
    DeleteSelected();
    count++;
}
$"Processed {count} files."
```

4.7 Buckling Equations

Buckling Equations

```
//Defines the equations for the flexural buckling of members without slender elements
(AISC 360 E3)
//After running the script, the elastic buckling stress, nominal stress, and nominal
compressive
//strength can be computed for a given effective length

//Elastic Buckling Stress
double Fe(double lc)
{
    return Math.PI * Math.PI * GetSectionProperty("E") / Math.Pow(lc /
GetSectionProperty("r2"), 2);
}

//Nominal Stress
double Fn(double lc)
{
    var fy = GetSectionProperty("Fy");
    var fe = Fe(lc);
    if(fy/fe <= 2.25)
    {
        return Math.Pow(0.658, fy/fe) * fy;
    }
    return 0.877 * fe;
}

//Nominal Compressive Strength
double Pn(double lc) => Fn(lc) * GetSectionProperty("Area");
```

5 Documentation

5.1 Reporting

ShapeBuilder offers both quick and custom-made reports. The two built-in reports, Basic and Full, are designed to get you what you typically need with the click of a button. You may also manually build up the report you want by adding and removing tables and graphics. After finishing your report, you can print the report or export it to a spreadsheet or any text-based program. Please note that currently there is no way to setup and save a custom report template. However, the active report inside of the project file will be saved and available upon reopening the project.

Modifying Reports

You may modify any report after it has been created in the following ways.

Width

Click on a graphic or table in the report and use the **Project Manager | Report Filter** to change the width of the image or table.

Order Number

Each table or graphic has an Order Number, which helps organize items in the report as they are automatically positioned to fit the page. The higher the number, the further down in the report the item will be located. Items with the same number are grouped together as much as possible. To move something to the end of the report, give it a large number. To make something first you can give it a negative number. Unless you give every single item a unique Order Number, you won't be able to control the exact order of items as they are positioned by size.

Add/Remove Items

You can add a table to the report by dragging one from the Available Tables list onto the report or the Included Tables list. You can remove any table or graphic image using the **Project Manager | Tables** control. Click the X next to the name of the Included Table.

Property Descriptions

You may optionally include the definitions of properties using the Show Descriptions check box. This is a report-wide setting that will automatically make tables full-width so that the definitions are easy to read.

Copy & Paste Images

The built-in report images may not be as clear or zoomed for optimal reporting. Use the Copy command to capture either graphic view, exactly the way you want to see it, switch to the **Report View** and Paste the image (you can paste any image from any product into your ShapeBuilder report).

Basic Report

The Basic Report contains a graphic from the active Sketch View or Analysis View. It also includes the simple and advanced results. The image placement depends on the aspect-ratio of the sketch view window. You may customize this report after it has generated to add, arrange, or remove items.

Full Report

The complete report includes a full page Sketch View and well as full-page graphical views of all the calculated stresses and functions. It also includes the simple and advanced results. You may customize this report after it has generated to add, arrange, or remove items

5.2 Exporting

Shapes can be exported to the IES Shape Database or to a DXF/DWG file and the results can be exported to a text file or copied into another program. Watch the Importing & Exporting video for an example of how to export shapes and results.

Exporting to DXF/DWG

Click **File | Export | Export to DXF/DWG** to saves the current shape (outlines and holes) as an CAD file. The file is written in inches.

Exporting Results

Click **File | Export | Export Results** to save the current simple and advanced analysis results into a tab-delimited text file. The Copy command will also place text results and graphics onto the Windows Clipboard that you can Paste or Paste Special into another program.

Exporting to the IES Shape Database

Click **File | Export | Export to IES Shape Database** to export the shape for later use in ShapeBuilder or other IES applications such as VisualAnalysis. The following provides details on the options provided in the Export to Database dialog box.

Shape Tree Location

Database: Define a name for your new database, or select an existing database location for your custom shape.

Category: Define a name for your new category, or select an existing category for your shape. It is a good idea to group shapes with similar properties or configurations into the same category.

Shape Name: Provide a descriptive name for your shape. This is how you will find it and the name that will appear in most reports.

Design Type (Shape Classification)

Analysis Blob is the standard Shape Type used for any shape that can be analyzed in VisualAnalysis but not designed. For design checks, choose the correct Shape Type from the drop-down menu and provide the required properties for the profile. IES does not recommend that you try to "fool" the system by selecting a Shape Type that is not accurate. Most built-up or custom shape profiles will not be shapes that VisualAnalysis can design. For example, a wide flange shape with a cover plate is not a wide flange shape and you will likely get unreasonable design checks for such a shape.

Shape Properties

Default Material: The material used for the shape when exporting.

Dimensions and other Required Properties: Based on the selected Shape Type, additional dimensions or other properties might be needed for the database. Depending on how your shape was created, ShapeBuilder need to provide additional values before continuing.

Optional Properties: You may optionally define "Shear Areas" for your shape, these are normally left at zero, but if you

wish to explicitly include shear deflections during an analysis in VisualAnalysis you must define these values. This is not a commonly used feature and requires the advanced level of VisualAnalysis as well as additional settings in that product.

Export Limitations

Database Categories: Categories can only contain one shape "design" type. Selecting an existing category dictates the current shape's type. If you would like to export this shape using a different type, select a different category or create a new category.

Composite Shapes: Shapes are exported as a single-material shape, with transformed section properties, and are seen in other IES products as a single shape with these transformed properties. You must define your shape parts with one material before the export will be permitted.

Cold-Formed Shapes: You cannot export cold-formed steel shapes from ShapeBuilder for design unity checks in VisualAnalysis. The only way to get custom cold-formed shapes into VisualAnalysis is to import a CFS file, as more data is required for design checks.

Tips To Get Design Checks in VisualAnalysis

Not all custom shapes can be designed in VisualAnalysis. In order to get design-checks your exported shapes must satisfy all of the following requirements:

1. Shape must match a **Shape Type** of existing designable shapes. Analysis blobs are not designable.
2. You must provide the **Required Properties** that ShapeBuilder has not calculated, if any.

5.3 References

1. **Manual of Steel Construction** and **ANSI/AISC 360-16** by the American Institute of Steel Construction, 2017.
2. Young, Warren C. and Richard G. Budynas. **Roark's Formulas for Stress and Strain**, 8th Edition, 2011, ISBN 978-0071742474.
3. Seaburg, Paul A. and Charles J. Carter. **Torsional Analysis of Structural Steel Members**. Steel Design Guide Series #9. 1997, American Institute of Steel Construction.
4. Murray, Noel W. **Introduction to the Theory of Thin-Walled Structures**. Clarendon Press - Oxford 1984, ISBN 0-19-856186-5.
5. Pilkey, Walter D. **Stress, Strain, and Structural Matrices**. John Wiley & Sons, Inc. 1994, ISBN 0-471-52746-7.
6. Davis, Raymond E., Francis S. Foote, James M. Anderson, and Edward M. Mikhail. **Surveying Theory and Practice**. McGraw-Hill, Inc. 1981, ISBN 0-07-015790-1.
7. Gruttmann, F., R. Sauer, and W. Wagner. **Shear Stresses in Prismatic Beams with Arbitrary Cross-Sections**. International Journal for Numerical Methods in Engineering No. 45, p. 865-889, 1999.
8. Gruttman, F. and W. Wagner. **Shear correction factors in Timoshenko's beam theory for arbitrary shaped cross-sections**. Computational Mechanics No. 27, p. 199-207, 2001.
9. Galambos, T. V. **Structural Stability of Steel Concepts and Applications for Structural Engineers**, Wiley, 2008, ISBN 978-0-470-03778-2
10. Trahair, N.S. and Nethercot, D.A. **Buckling of Laterally or Torsionally Restrained Beams**, Journal of the Engineering Mechanics Division, ASCE, 99, No. EM4, pp. 773-91, 1984