# AxisVM v8.0 Features and Benefits

AxisVM is a 3D structural analysis system based on the finite element method. It allows total building calculation inclusive of different materials and structural elements like slabs, columns, beams, ribs and more. Building geometry can be created with the CAD modeler or imported; loading can be assigned by building component, by specified locations, or globally. Variable conditions such as references or supports can be assigned or modified as required. Fast calculation results allow the engineer to quickly review the building structural design, analyze and make changes graphically or by editing the model table, and then quickly recalculate to see the results of the modified parameters.

## Integrated Building Model

These features allow the engineer to work with an entire structure and reduce the time requirements and error potential of separate analysis processes. Independent preprocessing of the model geometry allows components to be assigned values according to the needs and requirements of the engineer, they are not restricted to pre-assigned building templates or limited to predetermined element values. Full graphic editing permits fast selection of model elements and intuitive dialog windows include the choices of all parameters for the selected element. Loads can be combined and values of the combinations assigned that enable the engineer to determine the optimum design value and the ultimate failure conditions.

| **Feature** | **Functions and Benefits** | **User Book Reference** |
|---|---|---|
| Graphic User Interface | Drawing the model in 2D or 3D perspective views using the multiple views feature makes 3D modeling easy. User transition from 2D experience to 3D competence is fast and accurate. Quickly make changes to your design and arrive at optimized models in the shortest time. | 3.0 |
| Element Definition | Model geometry parts can be selected individually or be assigned to domains and material properties (concrete, steel, custom defined, etc.), finite element types (beam, membrane, plate, etc.) defined, constraints and conditions determined, relationships between elements created, and loading assigned as desired. | 3.9 |
| Postprocessing | Results by parts, types, elevations, or materials allow the engineer to compare the design to the building requirements. | 5.0 |
| Design | The results of the model are based on code specific design requirements, permitting a final design to meet the parameters of the code and optimize the structure. | 5.4 |
| Table Browser | All details of the model can be organized by element, material, cross section, domain, load or result. Editing of individual items is reflected in the model in real time. | 1.7 |
| Report Maker | The Report Maker can include full project details including images and tables and is compatible with MS Office programs. Data is automatically updated as model changes. Standard report requirements can be saved as a template reducing the time required to create and submit engineering details. | 1.8 |

## Graphic User Interface

These features enable fast and accurate modeling of the geometry of simple or complex structures, individual components, and cross sections of building materials.

| Feature | Benefit | User Book Reference |
|---|---|---|
| Designed for Windows | Most of the native Windows shortcuts, CTRL, ALT and keyboard commands are available as well as the use of Hot Keys. Drop down menus work like most Windows programs | |
| Units and Formats Settings | Predefined units of measure and values of properties such as SI Units or US units are included. User defined settings allow for the model to be valued in ft/in and the loadings or material settings to be an SI value. During editing or upon completion of an analysis, a model can be converted to/from one Unit/Format to another to view the information as needed by multiple users. | 1.7 |
| Grid Settings | Most models can use the default settings associated with the global Units/Formats setting, but if a desired geometry requires unique conditions or precision, then the grid, mouse snap, intersections, and coordinate types or other values can be specified. | 1.9.12.1 |
| User Preferences | Change the way and times that you save the model, create specific undo/redo values, change the appearance of the screen, the types and sizes of fonts on the screen or in the reports, or set the report language as independent from the license language. | 2.3.8 |
| Drawing Guidelines | A guideline ruler can be turned on and off in the model to assist the user to edit to the correct coordinates and allows X, Y, X-Y or oblique settings. Difficult drawings are easier to enter and user errors minimized. | 1.9.5 |
| Line Drawing Constraints | Lines can be limited to perpendicular or parallel conditions that ensure the speed and accuracy of the graphical input. Planes can be restrained to the user choice. | 1.9.6 |
| Dimensions and Text Labels | Dimensions of lines and angles can be added to the model and their display turned on/off as needed. The values and conditions for the dimensions are defined by the user, allowing for fast and accurate modeling. | 1.9.7 |
| Defining Parts of the Model | Any part of the model can be uniquely identified, assigned a name and displayed, allowing a single component or single type of component to be edited independent of the rest of the model. Identifying only floors or walls or columns makes modeling faster, and a part can also be assigned to a specialist work group enhancing collaboration success. All of the details, i.e. materials, cross-sections, thickness, and loads are included in the part. | 1.9.8 |
| Section Lines and Planes | The model can be graphically sectioned by lines or planes that allow analysis results to be displayed in these sections. The user can decide on the critical areas and view only the results from his defined section, allowing the display results to be more conveniently displayed. | 1.9.9 |

| | | |
|---|---|---|
| Find | The Find tool will identify any element type, i.e. beam, domain, support, or link and graphically highlight the found item in the model. Using the find tool with the Table Browser permits the engineer a component-by-component edit capability ensuring a model that meets the exact conditions necessary for an accurate structural analysis. | 1.9.10 |
| Model Display Options | Users can turn on/off the symbols, labels and info windows of each type of model component so editing and review of separate modeling steps can be further enhanced. Display options work on the whole model or on only a defined part. As an example, the engineer can view only a wall with loads, and then turn on/off the line support to verify the model is defined as needed, | 1.9.11 |
| Multi Windowing | Perhaps the best tool for modeling graphically with accuracy. Users that normally use 2D software can turn on the X-Y (Top View), Z-X (Front View) and Z-Y (Side View) and still have a 3D perspective view available upon demand, or see all four views at the same time. Windows can be split horizontally or vertically, each with its own display settings, allowing multiple perspective views. Structural modeling in 3D has never been easier. Editing in one window reflects in real time the display in other windows. | 2.5.2 |
| Geometry Entities | Straight lines, polylines, rectangles and arcs draw simple shapes, but in a 3D combination allow the most complex of models to be designed. | 3.8.2- |
| Mesh Refinement Tools | Horizontal, Vertical, Line, Quad and Triangle Divisions can be applied to the surfaces and lines of a model. Options to set the number and type of divisions allow user control. | 3.8.4- |
| Geometry Editing | Models or parts can be moved, copied, rotated, mirrored or resized with multiple options as to the conditions of each action. The user needs only to define a geometry component once, and then clone the component and all its associated properties to make modeling fast and eliminate repetitive steps. | 3.8.9- |
| Geometry Import | An existing DXF, ArchiCAD or IFC design can be imported as a new model or added to an existing model. The user can select to import only to the background layer, or to include nodes and lines in the model. Intuitive import features save countless modeling hours and enhance accuracy and allow integration with other user software. The user can turn on and off the imported background layer. | 2.1.6 |

## Element Definition

In addition to a large selection of material values and cross-sections, AxisVM provides various types of finite elements with user defined properties. A structural model can be defined by creating a mesh of user defined elements to the geometry of the model to assure that the analysis represents the most accurate computer aided engineering representation of the structure. The Structural model can include different element types allowing complex structures to be accurately modeled.

| | | |
|---|---|---|
| Finite Elements | The user-defined properties for each type of finite element permit the parameters determined by the engineer to be taken into account in the analysis. Depending on the type of finite element, the user can define the following properties: | 3.9 |
| Material Properties | The material library includes most common European building materials and their associated properties. The property values in the default library can be user modified as well, allowing for the material data to match the building products that are available in the local market. | 3.9.1 |
| Custom Materials | The user can define new materials and properties and add them to the library that saves them for future use. Specific manufacturer materials can thus be added. Custom materials can be defined with isotropic or orthotropic parameters and the associated material values assigned for each. This allows the user to select customer specified materials, which may be unique to the project. In addition, when modeling an existing structure it can be useful to assign a specific expected value to the building's existing material, especially if the material properties were unknown at the time of construction. | 3.9.1 |
| Cross-Sections | The included library of common shapes can be user edited, or new shapes added. Shapes can be drawn graphically or the details imported from a dbase format supplied by an industry association, customer, or component manufacturer. Modeling structures for industrial or plant applications are simplified and the versatility of the engineering applications enhanced. | 3.9.2 |
| References | In addition to the assignment of automatic references, which can be changed or deleted, the user can define reference points, vectors, axes, or planes. The proper assignment of the reference to the finite elements will result in smooth internal force and stress diagrams that are easy to understand and evaluate. | 3.9.3 |

| Finite element | Material | Cross-section | Ref | Stiffness | Surface |
|---|---|---|---|---|---|
| Truss | • | • | o | | |
| Beam | • | • | • | o | |
| Rib | • | • | o | | |
| Membrane | • | | • | | • |
| Plate | • | | • | | • |
| Shell | • | | • | | • |
| Support | | | • | • | |
| Rigid | | | | | |
| Spring | | | o | • | |
| Gap | | | | • | |
| Link | | | | • | |

| | | |
|---|---|---|
| Nodal Supports | The user can define nodal support in a global reference or relative direction, input the stiffness values, and assign nonlinear resistance values. Each node's support conditions are listed in the table browser and can be edited or found as needed for complete model control by the engineer. Further support stiffness values and end releases can be defined for columns. The engineer can confidently and accurately model buildings that include combinations of steel and concrete materials. | 3.9.6 |
| Line/Surface Supports | Beams, ribs or surface edges can be assigned Winkler type support elements, with stiffness values user defined. Nonlinear behavior can be specified. | 3.9.7 |
| Special Elements | Rigid, spring, gap, and link elements allow highly accurate modeling of connections and supports. Depending on the element, the stiffness, resistance, mode of action (tension or compression), or reference orientation, can be defined for each type. Membrane-beam connections, bolted connections, plate contacts, girder-purlin connections are a few examples of how the engineer can use these special elements for accurate modeling. | 3.9.9- |
| Nodal Degrees of Freedom | Translational and rotational, and free or fixed constraints can be defined for nodes. Default choices for typical structures assign predefined settings appropriate for plane truss girders, plane frames, grillages, membranes, and plates. The model will then include the least number of unknowns allowing for a faster analysis. | 3.9.13 |
| Domains | Domains can be defined for floors, walls and any other complex surface geometry and multiple domains can be used in a structural model. A domain can further be defined as a part, and the engineer can model and review the analysis on a domain-to-domain basis. Holes and meshes can be specified by the user to obtain the geometric accuracy needed for the analysis result. | 3.9.14 |
| Loads | Loads can be assigned to nodes, lines, domains and surfaces. Force, fluid, seismic, thermal load and other types can be defined for modeling residential, commercial or institutional buildings, industrial structures, water tanks, towers, etc. are available to the user. | 3.10. |
| Load combinations | Load combinations can be defined by the engineer that can include one or more load cases representing different loading conditions. Load groups can be defined as permanent, incidental or exceptional. The load cases in each load combination are taken into account through the assigned load factors. Dead loads, live loads, wind, snow, ice, crane runway, earthquake, support settlement, explosion and other engineer requested static loads can be modeled The engineer can see the analysis results for an entire structure, and by using the parts and section lines and surface tools, see the results on individual elements of the structure. The engineer has a macro-analysis tool with a micro-analysis viewer. | 3.10.1- |

## Postprocessing

AxisVM performs static, vibration and buckling analysis. The structural engineer can study the behavior of the structure by viewing the graphic displays and considering the tables of results. In AxisVM, the user does not have to specify a limit for the analysis calculations available. The processing engine in the software is fast and the results obtained on most Pentium III or better CPUs are available within a short time.

| | | |
|---|---|---|
| Result Display Options | The user can view any of the analysis results by selecting the type of analysis, envelope or critical values with Min/Max definition, undeformed or deformed shapes; diagram, section line, isoline or isosurface display. The engineer can view the results in 2D or 3D. | 5.1 |
| Result Labels | The actual result of the load combinations and chosen node, line or surface element result can be displayed. Labels of these results can be assigned to the corresponding entities. The engineer can see the graphic view and note the result label detail concurrently. Changing from view to view, component to component, or using multiple windows to view the results of different result components allow the engineer to quickly grasp the behavior of the structure. | 5.1- |
| Result Components | Displacements, internal forces, stresses and influence lines are result components that the user can select. A min/max value can be requested for each of these components and the location of these min/max conditions will be highlighted graphically. The user can zoom on a specific area and then move to the elements tab to inspect the possible factors that influenced the result. | 5.1.5- |
| Vibration/Buckling Analysis | AxisVM will calculate and display the mode/buckling shapes, and frequencies/critical load parameters. Animated result displays provide enhanced comprehension of the structural behavior. | 5.2- |

## Design

Concrete plate, beam/column or steel beam/column design based on the results of the analysis, integrate the same Postprocessing features. Modifications based on the design results can be made directly on the model for an interactive design process.

| | | |
|---|---|---|
| Concrete Surface Reinforcement | Reinforcement for membranes, plates and shells is based on the 3rd stress condition. The engineer can specify the type of concrete, the steel rebar properties, and the top and bottom concrete cover. Calculations are related to the load cases and combinations. The engineer has access to an accurate and fast design process. | 5.4.1 |
| Crack Opening Calculation | Engineers can verify that the potential crack opening of the concrete element meets the requirements of the design code or user. | 5.4.2.1 |
| Nonlinear plate deflection | The nonlinear calculation follows up on the static linear analysis utilizing moment-curvature diagrams of the cracked cross sections. More accurate prediction of deflection values is produced. | 5.4.2.2 |
| Column Reinforcement | Graphic and tabular numeric values of the load-moment diagram as compared to the design internal forces produces a reinforcement value that is safe and in compliance with the selected design code. | 5.4.3 |

| | | |
|---|---|---|
| | The parameters for calculation of the load-moment strength interaction diagram can be user defined. Display modes for the interaction diagram can be set by the user in the column check mode. | |
| Beam Reinforcement | Longitudinal reinforcement about the y or z-axis and the spacing of vertical stirrups considering the shear forces are included in the beam design function. The user can select the cross-section; the concrete and steel rebar materials and the concrete cover for the top and bottom. Tension, compression and minimum reinforcement is displayed. | 5.4.4.2 |
| Steel Beam/Column and Bolted Joint Design | A selection of cross-section shapes is available for design, and the program calculates the member resistances. The user can input the flexural, lateral-torsional, web shear/buckling and other parameters. The results are displayed graphically and in the results tables. The user can input the joint geometry for plates, bolt types, rows and spacing, and specify the materials. The efficiency diagrams based on the obtained results of the design, allow the engineer to easily identify the critical parts of the members. | 5.5.1 |

## Table Browser

The complete model details, locations and values of the materials, cross-sections, references, nodes, finite elements, domains …and the complete result details, locations and values for the displacements, internal forces and stresses are included in the Table browser. The engineer can view the model graphically and review the components and results of the model on an item-by-item basis. Complete understanding of the model and the analysis results is easy. .

| | | |
|---|---|---|
| Selective Results | Model data and result data can be limited by selection of a part or other current graphic selection. The engineer can review only what he needs to review. | 1.7 |
| Export | The data from a selected table can be exported in Dbase, HTML, TXT or RTP format. These exported files can be incorporated into any compatible software, database or report. Times consuming reporting tasks are eliminated. | 3.10. |

## Report Maker

The integrated report maker provides the engineer with the ability to create custom reports by selecting only those model data tables and graphic results needed. Users can define a style template required by regulatory authorities or contractor, saving time and enhancing cooperation.

| | | |
|---|---|---|
| Auto Update | Model changes automatically update tables exported from the Table Browser and incorporated in the report, are automatically updated with the updated model data and results. The user has confidence that the report submitted is the most recent and accurate. | 1.8 |
| Multiple Saving/Embedded Tables/Images | The user can see a result table or graphic image and simply one-button click to save it to the report maker. Tables and graphic views from any step of the modeling process can be saved to the report maker. Benchmark increments like post geometry import, final geometry design, definitions of finite elements, loading conditions, and final results are all available to save as result tables and images. And of course, user defined part or section views and their | 1.8.7 |

| | | |
|---|---|---|
| | independent parameters or results can be saved too. The engineer can build a report that includes the tables and images from every important step as he finishes it, and can later edit the final report to reflect only the values needed. The report can include bitmap images (.BMP, .JPG) and Windows Metafiles (.WMF) | |