

4.12 Mesh



Clicking on the *Mesh* tab the meshing toolbar becomes available with mesh generation for line elements and domains, mesh refinement functions and a finite element shape checking.

4.12.1 Mesh generation

Automatic detection of overlapping lines and missing intersections during meshing reduces the errors in model geometry.

Support of multiple core processors can reduce the time of meshing.

4.12.1.1 Meshing of line elements



Finite element analysis uses linear elements with constant cross-section so arced and variable cross-section (tapered) line elements must be divided into parts. This is called line element meshing. The accuracy of the solution depends on the mesh density.

This mesh can be removed or modified just like a domain mesh. Removing a mesh does not delete loads and properties assigned to the line element.

A mesh can also be defined for linear elements with constant cross-section. It is useful in nonlinear or vibration analysis when it is required to divide line elements to achieve a higher accuracy.

Mesh parameters for line elements

Mesh generation can be performed according to different criteria:

Maximum deviation from arc: Chord height cannot exceed the value specified.

Maximum element size: Length of the mesh lines cannot exceed the value specified.

Division into N segments: Line elements are divided into N parts.

By angle: Central angle of arced mesh segments cannot exceed the value specified.

4.12.1.2 Meshing of domains



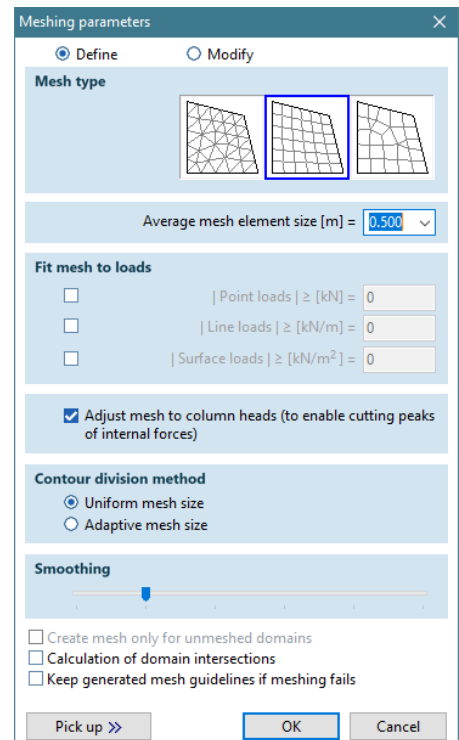
A mesh of triangular surface elements can be generated on the selected domains by specifying an average surface element side length for the mesh. Meshing will take into account all the holes, internal lines and points of the domain. Meshes optionally can follow loads above a certain intensity or be adjusted to column heads to enable cutting peaks of internal forces.

Mesh type The mesh can be a triangle mesh, a quadrangle mesh or a mixed mesh, in which most of the elements are quadrilateral with some triangles.

If lines of the domain outline including holes and internal lines can be divided into quadrangles and the quadrangle mesh is selected a better-quality parametric mesh is generated.

Mesh size An average mesh element size can be specified. The actual mesh can contain smaller and larger elements as well.

Fit mesh to loads Meshes will follow checked loads if load intensity exceeds the value specified. Point loads will create mesh nodes, line loads will create mesh lines.



Adjust mesh to column heads The mesh must be properly adjusted to column heads to prepare cutting peaks of internal forces. Turning this option on automatically fits the mesh according to the cross-section geometry of connecting columns. All beams joining to the slab at an angle greater than 45° are identified as columns. This option must be set to enable the *Cut peaks of internal forces over columns* option of the *Display Parameters* dialog. See [6.1.15 Surface element internal forces](#).

Contour division method *Uniform mesh size*
Domain boundaries and inner lines will be divided according to the mesh size to ensure the given element size.

Adaptive mesh size

Adaptive meshing follows domain geometry and refine the mesh by reducing element size wherever it is necessary.

Smoothing Track bar controls the smoothing of the mesh. Smoothing slows down mesh generation a bit. Moving the handle to the left end sets minimum smoothing and fast processing while the right end sets maximum smoothing with slower processing. The result of smoothing depends on domain geometry and other mesh parameters, so setting higher smoothing does not necessarily result in a better mesh quality.

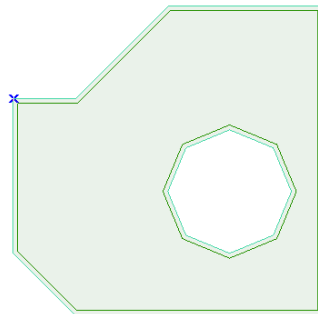
If *Create mesh only for unmeshed domains* is checked no mesh will be created for domains already meshed.

If *Calculation of domain intersections* is turned on domain intersections are automatically calculated before meshing.

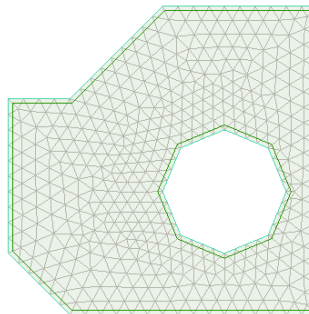
The progress of the mesh generation process can be monitored in a window, and can be canceled any time with the *Abort* button.

The mesh generator uses only the end points of beam elements that are in the plane of the domain, and disregards their corresponding line segments. Rib elements are incorporated with their line segments because they can be defined on surface edges as well.

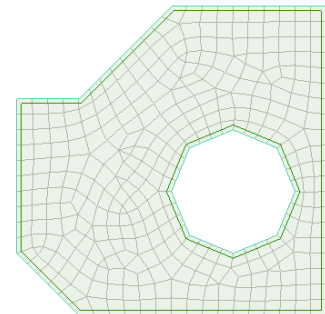
If there are existing quadrilateral or triangular meshes within the domain, the mesh generator will not change these meshes, and will integrate them in the new mesh.



Before meshing



Triangle mesh



Quadrangle mesh with triangles

☞ If a mesh is generated over an existing domain mesh (with a different average element side length), the new mesh will replace the existing one.

4.12.1.3 Meshing of soil modeling domains



The program generates a volume mesh with the given average horizontal and vertical mesh sizes for the selected soil modeling domains considering all the holes, nodes and lines inside the domains.

Choose the type of the generated solid elements from the *Geometry* panel. The shape of the generated solid elements can be extruded triangle (wedge), extruded quadrilateral (hexahedron), a mixture of the two or tetrahedron.

The solid element mesh is created by extruding the surface mesh vertically. In the case of tetrahedral meshing, the size and number of tetrahedra are dynamically adapted to the local layer thickness. If some layers are thinning to zero, the first three meshing methods may fail. In these cases, choose tetrahedral meshing.

Average mesh element size is the average element size of the planar mesh generated for the soil modeling domain.

Average vertical size is the average vertical element size of the solid elements. The actual vertical sizes also depend on the thickness of soil layers obtained from the stratigraphic model.

The generated mesh consists of solid elements, the total depth of the model is the largest of the *d* minimum values of the contiguous soil modelling domains. See 4.9.9 Soil modeling domain (SOIL module).

4.12.2 Mesh refinement

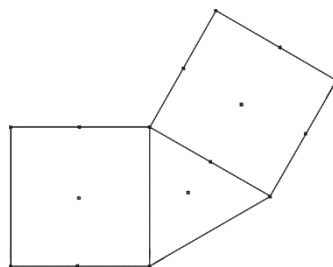


Lets you refine the finite element mesh of surfaces. The elements in the refined mesh have the same properties (material, cross-section / thickness, references, etc.) as those in the coarse mesh.

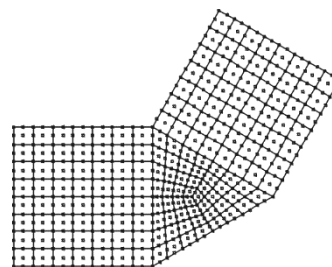
You have to manually set the nodal degrees of freedom of the newly generated mesh that were not set automatically during the process of mesh generation.

The following options are available:

Uniform



Before mesh refinement



After mesh refinement

Lets you refine the entire selected mesh. You must specify the maximum side length of a surface element in the refined mesh.

Bisection

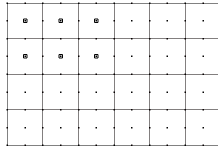
Lets you refine the selected mesh by bisecting the elements as shown in the figure.



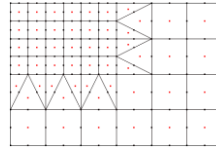
Quadrilateral element



Triangular element



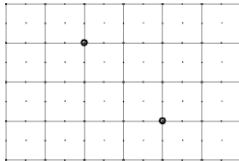
before



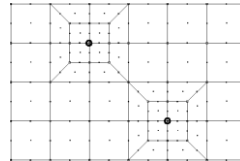
after

Node relative

Lets you refine the mesh around the selected nodes (locally around columns, nodal supports). You must specify a division ratio (0.2-0.8). The command refines the mesh dividing the elements connected to the respective nodes by the defined ratio.



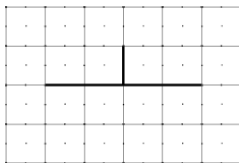
before



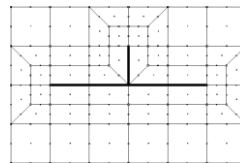
after

Edge relative

Lets you refine the mesh along the selected edges (locally along edge supports / loads). You must specify a division ratio (0.2-0.8). The command refines the mesh dividing the elements connected to the respective edges by the defined ratio.



before



after

4.12.3 Checking finite elements



Program checks the minimum angle of surface finite elements (α).

A triangular finite element is distorted if $\alpha \leq 15$.

A quadrilateral finite element is distorted if $\alpha \leq 30$.

4.12.4 Select free edges



This command selects all edges connected to one surface element only. This selection helps to find edges where the meshes of two domains are not connected due some editing problem.

4.12.5 Delete all meshes



Deletes all meshes from line elements



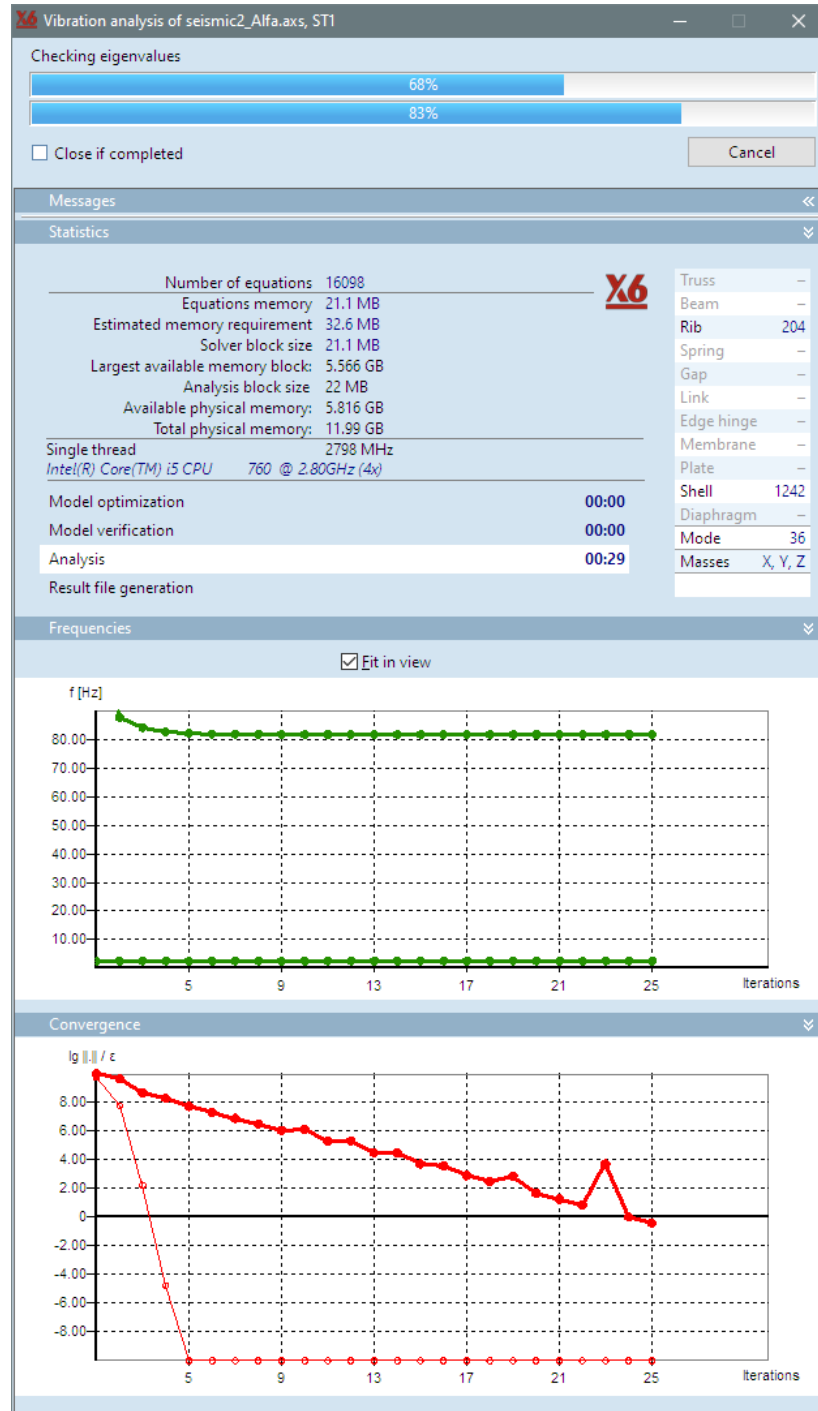
Deletes all domain meshes from domains



Deletes all (line or domain) meshes of the model.

5. Analysis

AXISVM lets you perform linear and nonlinear static, linear and nonlinear dynamic, vibration and buckling analysis. It implements an object-oriented architecture for the finite element method. The instructions included in this User's Manual assume a preliminary knowledge of the finite element method and experience in modeling. Note that the finite element analysis is only a tool, not a replacement for engineering judgment.



Each analysis consists of the following steps:

- 1) Model optimization
- 2) Model verification
- 3) Performing the analysis
- 4) Result file generation

Details of the analysis can be displayed by expanding one or more category panels. The *Messages* panel shows the analysis message log. The *Statistics* panel shows memory requirements, hardware information, model details and calculation times.

Special categories:

Nonlinear analysis: *Tracking* displays the movements of the tracked node. *Convergence* shows the convergence of the iteration process.

Vibration analysis: *Frequencies* displays how the frequencies converge. *Convergence* shows the convergence process.

Buckling analysis: *Eigenvalues* displays how the eigenvalues converge, *Convergence* shows the convergence process.

Dynamic analysis: *Time steps* displays the movement of the tracked node, *Convergence* shows the convergence process.



Parameters of the latest analysis is saved into the model file and can be studied in the Model Info dialog. See [2.16.20 Model info](#)

Model optimization

To reduce analysis time and memory footprint AXISVM optimizes node order. If the total number of degrees of freedom is over 1000, it creates an internal three-dimensional graph from the model geometry and begins to partition the system of equations using the substructure method. The system is stored as a sparse matrix. The parameters of the optimized system of equations appear only at the end of this process. This process results in the smallest memory footprint and fastest calculation time but it assumes that the biggest block fits into the available memory. If it doesn't, AXISVM stores the system as a band matrix and begins to reduce the bandwidth of the system by iterative node renumbering. If the two longest rows fit into the available memory the system can be solved. Changes in the memory requirements for the band matrix is displayed real-time. The duration of the optimization process and the final memory footprint depends on the size of the system and the available memory.



The system of equations can be solved the most efficiently if the whole system fits into the physical memory. If the system does not fit into the physical memory but its largest block does, the running time will be moderate.

If the largest block does not fit into the physical memory, the necessary disk operations can slow down the solution considerably.

Model verification

The input data is verified in the first step. If an Error is found a warning message is displayed and you can then decide whether to cancel or continue the analysis

Performing the analysis

AXISVM displays the evolution of the solution process by two progress bars. The bar on the top displays the current step performed, while the other displays the overall progress of the analysis process.

The equilibrium equations in the direction of constrained degrees of freedom are not included in the system of equations. Therefore to obtain support reactions you must model the support conditions using support elements.

The Cholesky method is applied to the solution of linear equilibrium equations.

The eigenvalue problems are solved with the Subspace Iteration method.

Error of the solution

Solution error is calculated from the solution of a load case with a known result. It is a good estimation of the order of errors in displacement results for other load cases.

Info palette shows this error as E(EQ).

If the value of E (Eq) is greater than 1E-06 the reliability of the computed results is questionable. It is expected, that the Error of the displacements is of the same order.

Result file generation

During the processing of the results the program sorts the results according to the original order of the nodes and prepares them to graphical display.

In the following chapters we 'll show the setting of the parameters of the each calculation methods.

5.1 Static analysis

The term *static* means that the load does not vary or the variation with the time can be safely ignored.

Linear static



Performs a linear static analysis. The term *linear* means that the computed response (displacement, internal force) is linearly related to the applied load.

All the load cases are solved in the analysis. Through the geometric linearity, it is assumed that the displacements remain within the limits of the small displacement theory. Through the material linearity, it is assumed that all materials and stiffness characteristics are linear-elastic. The materials assigned to surface elements can be orthotropic.

☞ See the description of the truss, gap, support, seismic isolator and spring elements in Chapter 4, on how to use these elements in a linear analysis.

The relative errors at the end of the iteration process appear in the info window.

$E(P)$: relative error of the force convergence
 $E(W)$: relative error of the work convergence
 $E(EQ)$: condition of the system of equations

Values indicating instability appear in red.

Nonlinear static



Performs a nonlinear static analysis. The term *nonlinear* means that the computed response (displacement, internal force) is nonlinearly related to the applied load. This can be due to the use of e.g., gap, link, non-linear support elements, line/surface elements with nonlinear material, or taking into account the geometric nonlinearity.

Select load cases or combinations in the tree view.

Nonlinear static analysis

Load cases

- All
 - Load cases
 - ST1
 - G
 - ST2
 - Load combinations
 - ULS
 - Co.#1 (ULS)

3 of 4

Solution control

☒ Force
 ☐ Tracked node

☐ Displacement

☐ Arc length (Riks)

☐ Pushover

☐ Equal increments

☒ Increment function

Convergence criteria

Maximum iterations:

☒ Displacement

☐ Force

☐ Work

☐ Use secant stiffness (only in appropriate cases)

Iteration control

☒ Automatic constraint for strain increments

☐ Constraint for displacement increments

Displacement [mm] =
 Rotation [rad] =

Use reinforcement in calculation

☒ Actual reinforcement

☒ Calculated reinforcement

☒ Creep

☒ Shrinkage

Nonlinearity

☒ Follow nonlinear behaviour of materials and finite elements

☒ Follow geometric nonlinearity of beams, trusses, ribs and shells

☒ Store last increment only

OK Cancel

AXISVM will perform nonlinear analysis for the selected load cases / combinations and shows a progress dialog.

Solution control

Force

When Force control is selected, the increments are applied as fractions of the loads (as one parameter load). It is possible to track the displacement of a node in a given direction. A graph of this displacement versus increments will be plotted during the analysis.

Displacement

When displacement control is selected, the increments are applied as fractions of the displacement component of the node specified.

Arc length (Riks)

When arc length control is selected, the increments are applied as fractions of the arc length on the force-displacement diagram of the selected degree of freedom of the selected node. This results in more displacement in the flatter parts of the characteristic and more force-like increments in the steeper parts. The target value must be defined as a displacement. The displacement in the first increment and the initial stiffness of the structure determines an initial arc length. AXISVM tries to keep the arc length close to this initial value in the subsequent increments using the algorithm proposed by Riks [44].

Pushover

Pushover control is a special type of displacement control that allows the use of a constant load case while having another parametric load case that is increased incrementally. This is essential for pushover analyses to model P- Δ effects appropriately.

After selecting pushover control, the top of the dialog changes to accommodate the drop-down boxes for parametric and constant load cases. See [4.10.26 Pushover loads – SE2 module](#) for details on load definition and recommended analysis settings.

Tracked node, Direction, maximum displacement

In case of displacement or pushover control a control node and a degree of freedom must be selected. Maximum displacement is the maximum allowed displacement of the control node in the given direction.

Number of increments

There are two methods to define the number of increments:

1. *Equal increments.* Specify the number of increments. The default value is 10. When highly nonlinear behavior is analyzed, you may specify a greater value in order to achieve convergence.
2. *Increment function.* Loads are not increasing in a linear way but follow a predefined function. Using an increment function it is possible to reduce the number of increments where the behaviour of the structure is linear and increase the number of increments where the behaviour is nonlinear. Saved functions can be reloaded, edited and saved under a new name (see [4.10.31 Dynamic loads \(for time-history analysis\) – DYN module](#)). Functions are saved into separate *.inc files in folder `c:\Users\[user name]\AppData\Roaming\AXISVM\[version number]\inc`.

Convergence criteria

Based on the convergence tolerances you specify, AXISVM will determine if the nonlinear solution has reached the required accuracy (convergence). Therefore it is important that the convergence tolerances to be set properly. During the iteration process, the norm of the unequilibrated load and/or of the iterational displacement increment vector must vanish (to approach zero).

Maximum iterations

You can set the maximum number of the iterations based on the specifics of your model, and of the incremental solution parameters. By default the value is set to 20. If the convergence is not achieved within the maximum number of iterations, no results will be obtained.

Displacement / Force / Work / Convergence criteria

In case of a nonlinear calculation, you can specify multiple criteria, in terms of load, displacement, and work, for monitoring the convergence of the nonlinear solution. At least one criteria has to be selected. The criteria expressed in terms of work can be adequate for most problems. However, you may encounter a small Error in your unequilibrated load while the Error in displacements is still large, or vice-versa.

Factors of convergence criteria has the following default values: 0.001 for displacements, 0.001 for force, and 0.000001 for work.

The relative errors at the end of the iteration process appear in the info window.

E(U): relative error of the displacement convergence
 E(P): relative error of the force convergence
 E(W): relative error of the work convergence
 E(EQ): condition of the system of equations

Use secant stiffness (in appropriate cases only)

If this option is selected beam end releases will be represented by their secant stiffness instead of tangent stiffness. This improves convergence but considerably slows down the calculation. It is recommended only if convergence cannot be achieved by increasing number of increments and iterations.

Iteration control

The weak convergence of a nonlinear calculation is often caused by nonlinear elements that have sharp changing of stiffness in their stress-strain characteristic. Examples of such elements are gaps, only tension or only compression trusses, supports having limit force and reinforced concrete structural elements. If the stiffness of several elements has to be change in the same increment, infinite loop of iterations can occur without reaching the convergence. To avoid this situation iteration control can be used. The implemented control methods constrain the size of the displacement increments in the critical iterations instead of decreasing the size of the load increment. With this strategy the convergence can be achieved but a lot of iterations would be necessary in the critical increment, because of the constrained displacement increments.

The program performs the following calculation steps in each iteration:

1. Calculates the displacement increment for each DOF caused by the unequilibrated loads.
2. Controls the DOF displacement increments if any of the control options is turned on.
3. Calculates further results...

The iteration control contains the following checks depending on the options selected.

Automatic constraint for strain increments

On the basis of the DOF displacement increments the program calculates the elemental strain increments. If excessive strain or stress increment or excessive changing in the stiffness occurs in any of the elements, the program calculates an allowed strain increment that does not cause any of the opreceding problems. After cheking all elements the program scales (decreases) the displacement increments so that excessive changing does not occur in any of the elements.

Constraint for displacement increments

With this option custom limit values can be defined for translational and rotational displacements. The program scales (decreases if necessary) the DOF displacement increments so that the maximum of them should be equal to the limit.

Use reinforcement in calculation

When analysing reinforced concrete plates, it is possible to take into account:

- the actual reinforcement;
- or the required amount of reinforcement calculated for a load case, load combination, envelope or critical combination;
- or the combination of previous two options.

When considering both the calculated and actual reinforcement (both checkboxes are active), the program determines the reinforcement amounts' envelope (maximum value) separately for each direction and reinforcement position. The program does not consider if the reinforcement position is specified differently for the actual reinforcement, and in the reinforcement parameters, only the amount of reinforcement matters.

This feature is available only for surfaces/domains. Only the actual reinforcement will be taken into account for reinforced concrete beams and columns.

Displacements and internal forces of reinforced concrete plates are calculated according to the moment-curvature diagram of the reinforced cross-section of the plate. These results show the actual plate deflection and forces in the plate (see [6.5.6 Nonlinear analysis of RC surfaces](#)).

In case of the analysis of reinforced concrete columns and beams, it is also possible to take the reinforcement into account. Internal forces compatible with strains are calculated through the integration of fiber stresses at Gauss integration points based on ϵ normal strains, κ_y and κ_z curvatures considering the actual reinforcement, the concrete and nonlinear material behaviour (see [6.5.5 Nonlinear analysis of reinforced concrete beam and column elements](#)).

The analysis can be performed either with consideration of creep/shrinkage or without. Further information related to the consideration of creep and shrinkage can be found in the above referred sections.

Nonlinearity

Follow nonlinear behaviour of materials and finite elements

This option is enabled if the model contains elements with nonlinear behaviour (e.g., tension-only trusses, compression-only supports, surface elements lying on soil elements) or elements with nonlinear material characteristics (trusses, beams, ribs, membranes, plates, shells). If left unchecked, all elements will respond in a linear way.

Follow geometric nonlinearity of beams, trusses, ribs and shells

The equilibrium is established with respect to the deformed line elements. Depending on the magnitude of displacement second or third order analysis is performed. The loads are conservative, they keep their initial direction during the deformation. Geometric nonlinearity can be taken into account only for truss, beam, rib and shell elements. If there are no elements with nonlinear characteristic in the model this option is checked by default. If the model contains elements with nonlinear characteristic this option is left unchecked but can be activated.

Stability warning messages

In geometric nonlinear calculations a stability problem may occur in some increments. It can be a local or global loss of stability or a snap-through phenomenon. Euler's critical load may also be exceeded in any beam. In such cases the following investigation procedure is recommended

- Checking of deformed shapes
- Buckling analysis to compare the critical load parameters and the load factors in the problematic increments.


A warning message appears in the info window identifying the increments concerned.

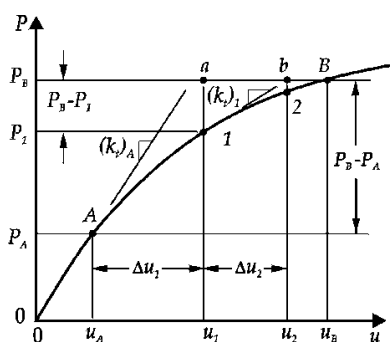
See [2.19.2 Info window](#)

 **Beam elements must be divided into at least four parts when geometric nonlinearity is taken into account.**

Store last increment only

Allows you to reduce the size of the results file when an incremental nonlinear analysis is performed with multiple increments (load or displacement) when just the results of the last increment are of interest to you. You can enable this checkbox when you do not need the results of previous increments.

 **You should disable this checkbox if you want to trace the load-displacement or other (nonlinear) response of the structure.**



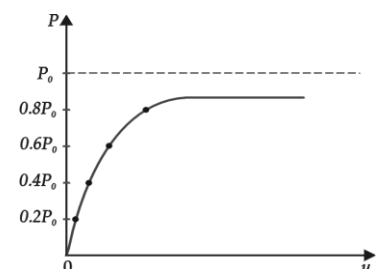
AXISVM applies a Newton-Raphson iteration technique to the iterative solution of each increment. The technique is known in different variants, depending on the update of the system (stiffness) matrix.

In AXISVM $n = 1$ (default), the system stiffness matrix is updated in each iteration. The method is known as the classical Newton-Raphson technique.

Displacement control

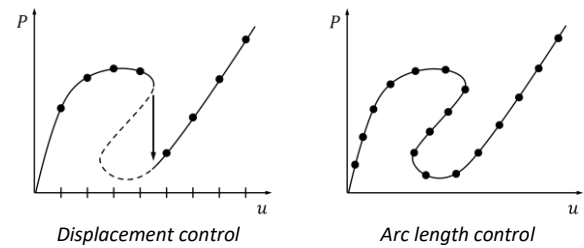
The so-called *snap-through* phenomenon cannot be analyzed with load controlled increments. You must apply a *displacement control* to pass through the peak points.

This figure shows a force control applied to a nonlinear system. The incremental solution fails in the 5th increment. To find the peak value of the load-displacement characteristics of the system, you must apply a displacement control technique.



Arc length control The so-called *snap-back* phenomenon cannot be analyzed even with displacement controlled increments. You must apply an *arc length control* to track the backward section of the characteristic. AXISVM uses the form of the arc length method proposed by Riks [44].

The figures show a force-displacement characteristic with a backward section. That part of the curve cannot be tracked with displacement control. But applying the arc length control the entire curve can be traversed with a nearly uniform step size.

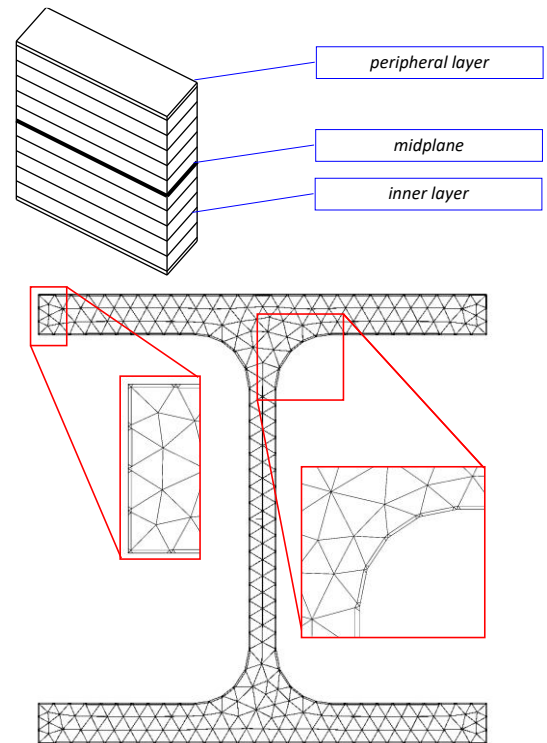


Finite elements with nonlinear material

Beam, rib and surface elements made of nonlinear (elastic or plastic) material are modelled with a discretized section model. Plates and shells are represented by layers, cross-section of beams and ribs are meshed. Stress distribution is determined using Navier's hypothesis (plane sections remain plane) and applying the nonlinear material model on each sub-element.

Stress components are the same as in the linear material model. For surface elements results are obtained in top, center and bottom plane, for beam and rib cross-sections in the stress points and along the outline.

To obtain accurate results in the extreme fiber the thickness of the layers is not equal. In case of plates and shells two thin peripheral layers are used, of which thickness is one tenth of an inner layer. The number of inner layers is ten. The cross-sectional mesh of beams and ribs also contains a thin peripheral layer, of which thickness is one tenth of the edge length of the inner elements. Stress results along the outline are obtained directly from the peripheral elements.



Seismic isolators See [3.1.16.2 Seismic isolators](#)

Compression-only connection between soil and base plate

If material nonlinearity is enabled, the connection between the soil and the surface elements lying on the soil is modeled with a compression-only connection. In the numerical model a surface-to-surface spring layer is inserted between the soil and the base plate. Its planar stiffness components K_x and K_y are linear, but its K_z stiffness component perpendicular to the surface is nonlinear and has a compression-only behaviour. The deformation of the spring layer in the active directions is negligible. The value of the stiffnesses:

$$K_x = K_y = 1000 \cdot \frac{E_{50}}{2(1+\mu)} ; \quad K_z = 1000 \cdot E_{50}$$

where E_{50} and μ are the secant modulus and the Poisson's ratio of the top soil layer. Due to the compression-only K_z component the surface is free to rise from the ground but the connection in the planar K_x and K_y directions still remains stiff.

☞ **Nonlinear link layer is created only under surface elements lying on the ground. No such layer is created under elements connected to the soil along a line or at a node.**