Section 1 Result

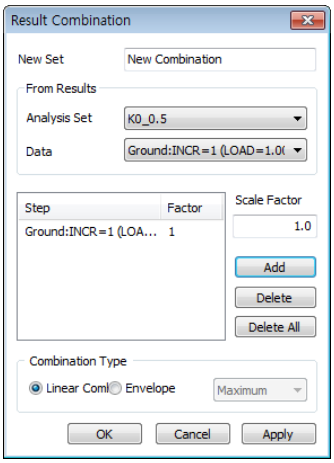
If the analysis has finished properly, check the results and collect, reduce the various numerical data needed for the actual design. The results, printed using the result combination and user defined formula, can be linearly combined to create a new result combination set.

1.1 Combination

Overview

If the analysis case is linear static analysis, the load sets in the case can be combined with various other case load sets. The load sets within a case can only be combined when the [Solve Each Load Set Independently] option is checked, and when both load sets have printed mesh set results.

▶Combination



Methodology

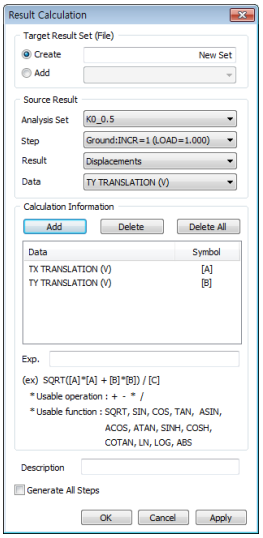
Define the new load set name and designate the analysis set and load results. The scale can be set when designating the load results.
There are two combination methods; the [Linear Combination] method, which linearly combines the analysis results, and the [Envelop] method, which depends on the value size. [Envelop] displays the maximum value, minimum value, maximum absolute value (no sign) and maximum absolute value (sign) for the analysis results of each load condition.

1.2 Calculation

▶Result calculation

Overview

Create combined results between particular result components.



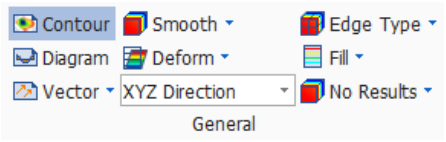
Methodology

Define the set name to create a new set, or perform additional computation on an existing selected file.
The letter case of the operator used in the formula can be mixed. The method of using is the same as that of a scientific calculator and the operation priority follows the mathematical laws.

Element	Description	Example use
(	Open parenthesis	-
)	Close parenthesis	-
+	Plus	-
-	Minus	-
*	Multiply	-
/	Divide	-
SQRT	√	<ex> √2 = SQRT(2)
SIN	Sine	Unit : Degree
COS	Cosine	Unit : Degree
TAN	Tangent	Unit : Degree
ASIN	Arc Sine	<ex> sin-1 (0.3)=ASIN(0.3)
ACOS	Arc Cosine	<ex> cos-1 (0.3)=ACOS(0.3)
ATAN	Arc Tangent	<ex> tan-1 (0.3)=ATAN(0.3)
EXP	Exponential	<ex> e0.3 =EXP(0.3)
SINH	Hyperbolic Sine	<ex> sinh(1)=SINH(1)
COSH	Hyperbolic Cosine	<ex> cosh(1)=COSH(1)
COTAN	Cosine / Sine	<ex> cotan(1)=COTAN(1)
LN	Natural logarithm	-
LOG	Common logarithm	-

Section 2 General

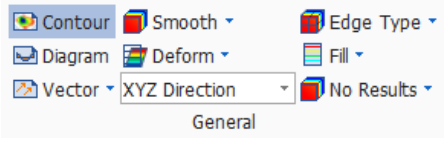
In the General section, the various graphic output processing methods for modeling analysis results, such as Contour, Vector, Diagram, Deform etc., are set.



2.1 Contour

Overview

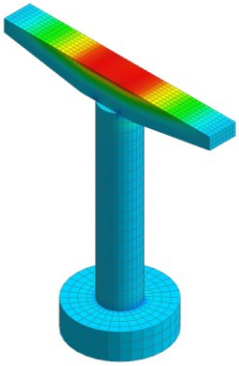
Display the displacement, reaction force, stress, strain size and direction of an activated element using contours.

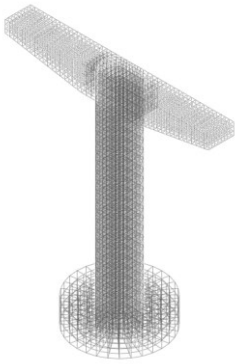


Methodology

Use Contour ON/OFF to decide whether to output the contour on the screen.

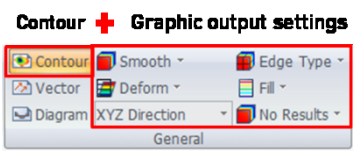
▶▶Contour OFF

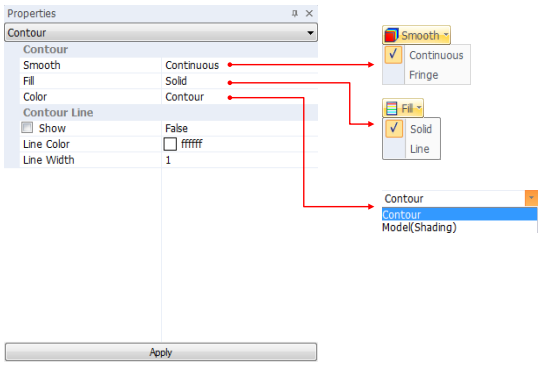




Contour property

The contours can be overlapped with the graphic display setting function to output more graphic results.

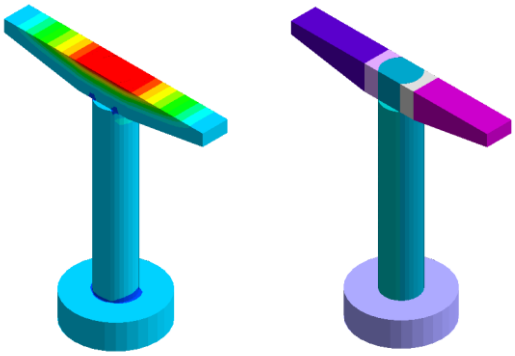




Color

Display the color of the selected results as the contour or model mesh color.

▶Contour
▶▶Model(Shading)



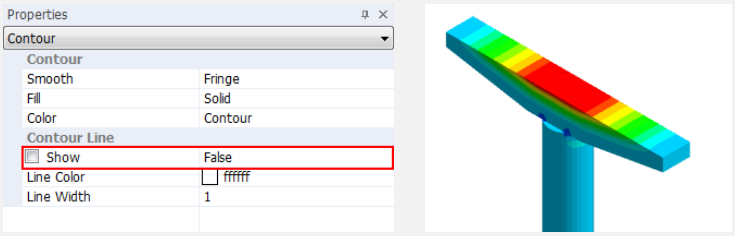
[Contour]
Plot the result color as the result contour color.

[Model (Shading)]
Plot the result color as the model mesh color. The analysis color can be displayed as result edge, deformed shape and fill type line draw, but not contour types.

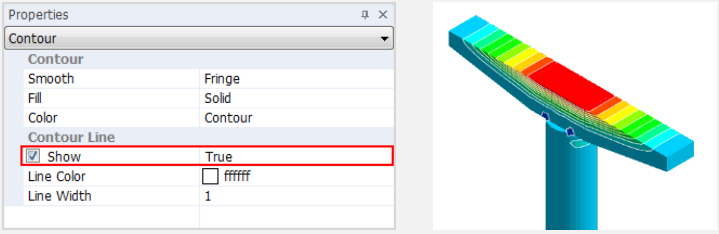
Tip

*** Setting the contour line**

The show line, color, thickness in the contour properties can be adjusted to clearly express the contour distribution.



< Show contour line - unchecked>

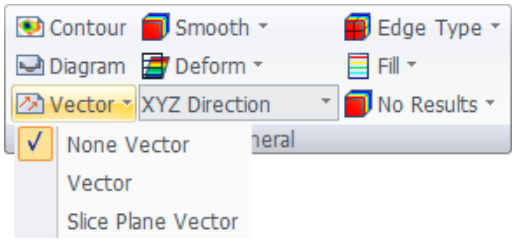


<Show contour line- checked>

2.2 Vector

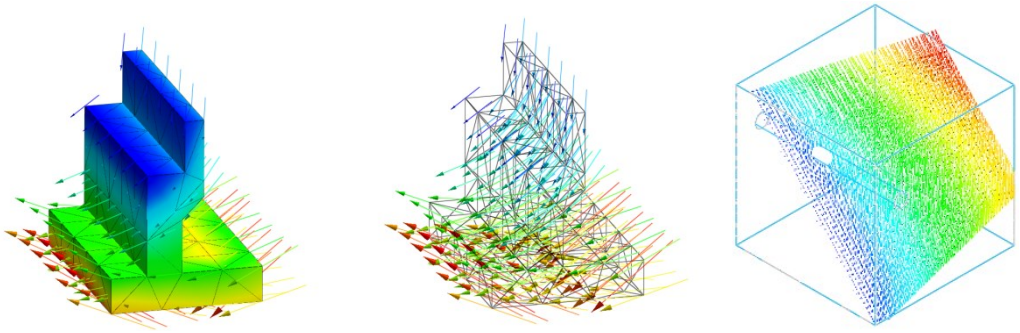
Overview

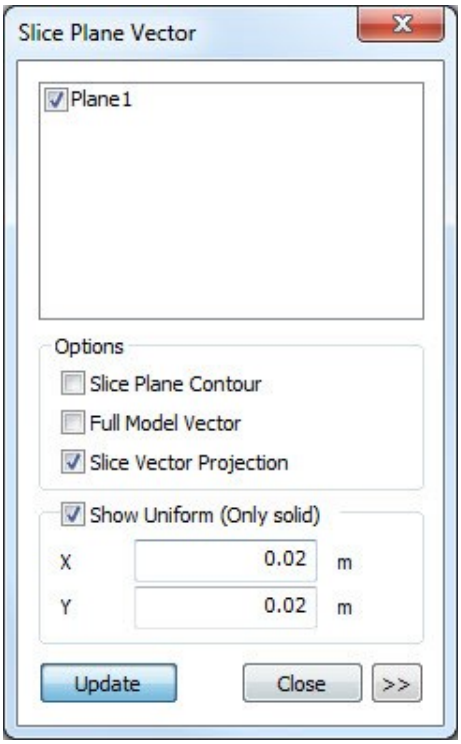
Display the size and direction of the selected displacement or reaction force component as a vector on each node.

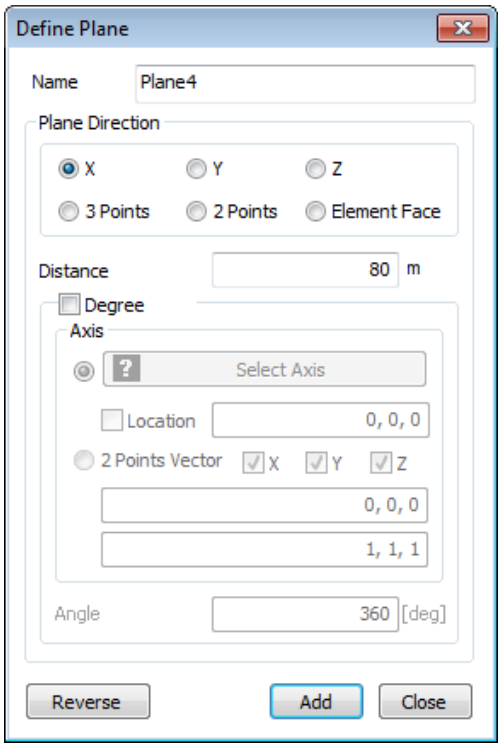


Details

▶Vector + Contour
▶▶Vector
▶▶▶Slice Plane Vector







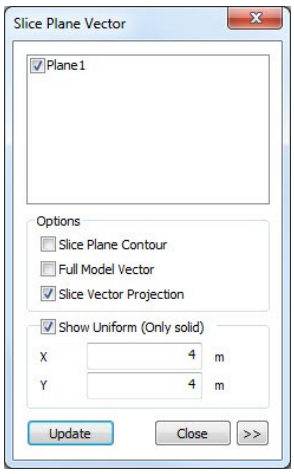
[Slice Plane Vector]
The vector plot is displayed on the cutting line/plane. The cutting line/plane definition method is same as the 'Clipping Plane'.

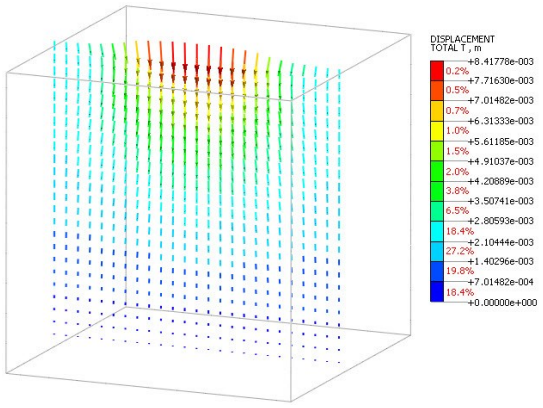
[Slice Plane Contour]
The contour and vector plot are displayed on the cutting line/plane at the same time.

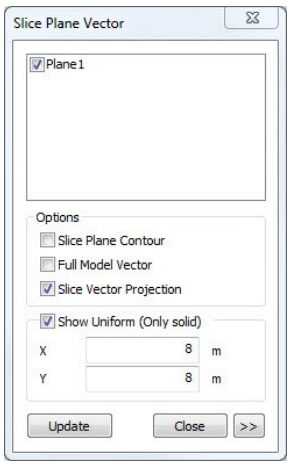
[Full Model Vector]
The vector plot of cutting line/plane and full model are displayed at the same time.

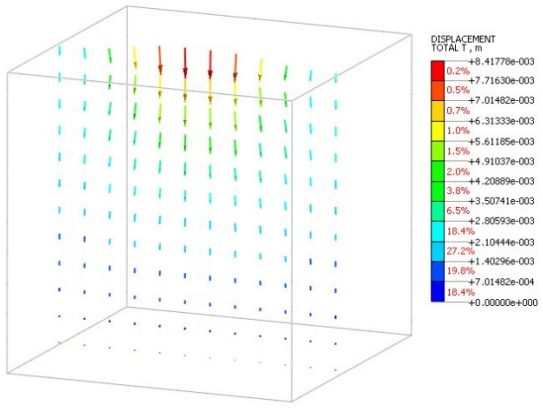
[Slice Vector Projection]
The vector plot is displayed by the projection of vertical direction on the cutting line/plane.

[Show Uniform (only Solid)]
This option controls display vector density on the screen. This option is model size sensitive - it is recommended to check dimensions of the model (geometry and/or element size) to insert reasonable input.





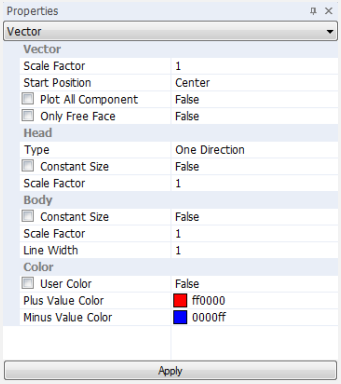




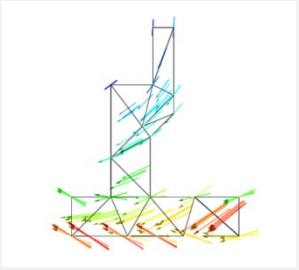
Tip

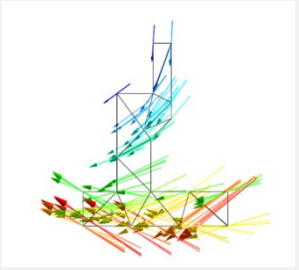
*** Setting the vector properties**

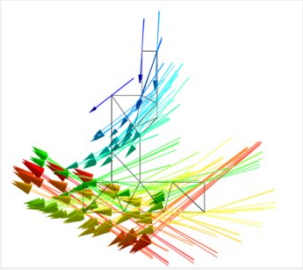
Use vectors to display the analysis results on the work screen. This option can be used to set how the vectors will be displayed.



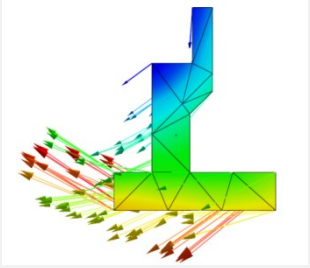
- Vector > Scale Factor
  Plot the scaled vector results.

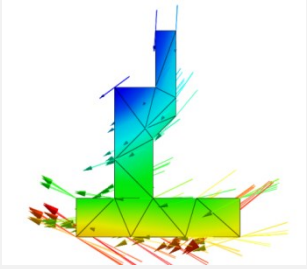

<Scale value = 2>

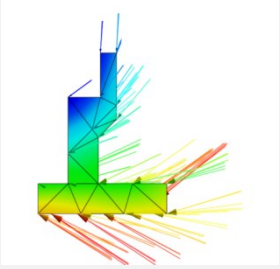

<Scale value = 5>


<Scale value = 10>

- Vector > Start Position
  Determine the arrow position.

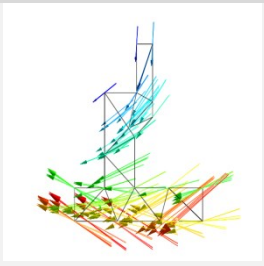
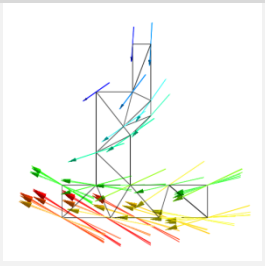

<Start position = Start>


<Start position = Center>


<Start position = End>

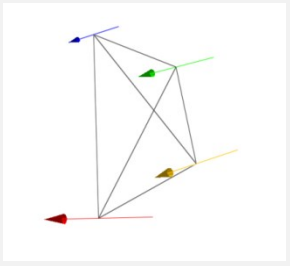
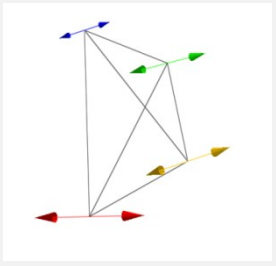
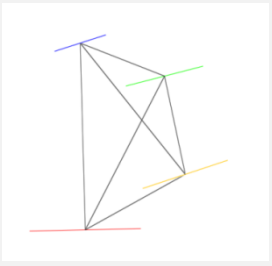
- Vector > Plot All Components
  When the principal stress vector of an element is output, both the maximum and minimum principal stresses are output if True.

- Vector > Only Free Face
  Only plot the arrows for nodes on the free face.

<Only free face Unchecked(False)> <Only free face checked(True)>

- Head > Type
  Determine the vector arrowhead direction. (One direction/ Two direction/ No direction)

<One direction> <Both direction> <No direction>

- Head > Constant Size
  Set all vector arrowheads to an equal size. (True/False)

- Head > Scale Factor
  Plot the scaled vector arrowhead. (Scale Value)

- Body > Constant Size
  Set all vector arrow body lengths to an equal size. (True/False)

- Body > Scale Factor
  Plot the scaled vector arrow body length. (Scale Value)

- Body > Line Width
  Plot the scaled vector arrow body thickness. (Scale Value)

- Color > User Color
  Plot the vector arrow using the user defined positive and negative colors. (True/False)

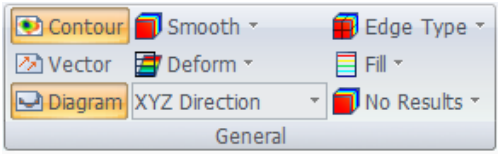
- Color > Plus Value Color
  When setting the user color, determine the positive value vector arrow color.

- Color > Minus Value Color
  When setting the user color, determine the negative value vector arrow color.

2.3 Diagram

Overview

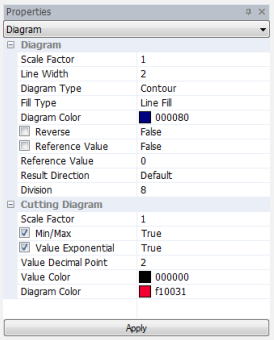
Display the selected 1D element results as a diagram.



Tip

* Setting the diagram properties

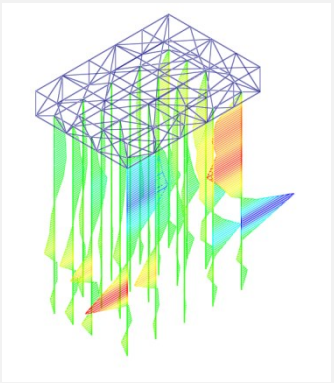
Specify the display settings for a 1D element result diagram.

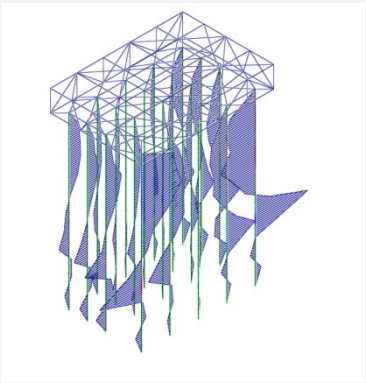


- Diagram > Scale Factor
  Plot the scaled member force or stress diagram of a 1D element.

- Diagram > Line Width
  Plot the scaled Diagram line thickness.

- Diagram > Diagram Type
  Select the diagram type that reflects the analysis results, using contours or a single color, and plot.

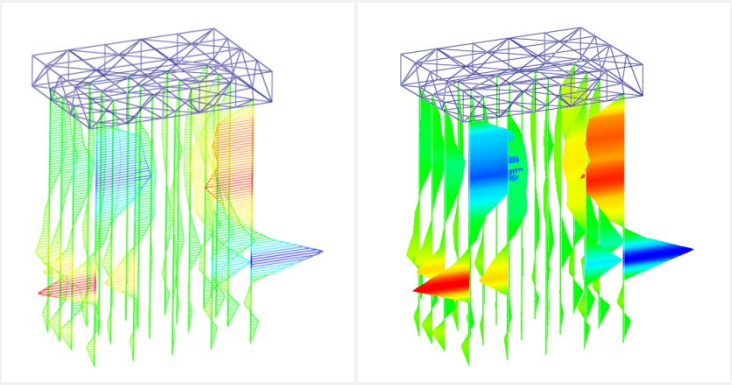

<Contour>


<Single color>

- Diagram > Diagram Color
  Define the color for the single color type diagram.

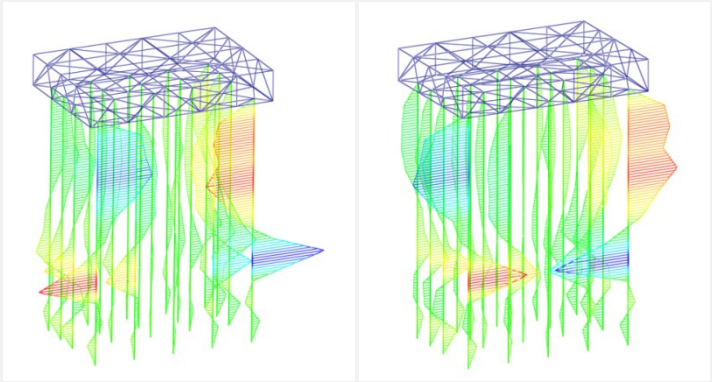
- Diagram > Fill Type

Plot the diagram using a fill type; either a line fill or solid fill.



<Line fill> <solid fill>

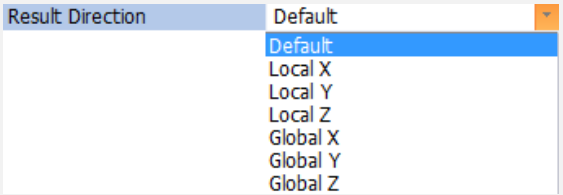
- Diagram > Reverse
  Reverse the direction of the diagram display and plot.

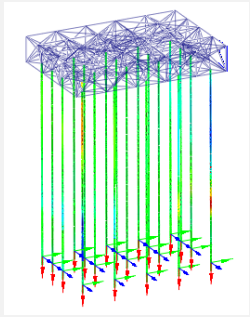


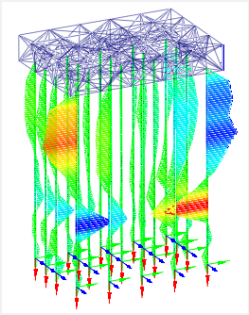
<Reverse(True)> <Revers(False)>

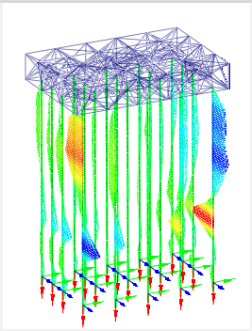
- Diagram > Reference Value
  Determine whether to display the diagram using a relative value, with respect to the reference value.

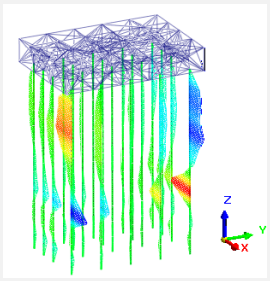
- Diagram > Result Direction
  Set the diagram display direction with reference to the element coordinate axis or GCS, and plot.

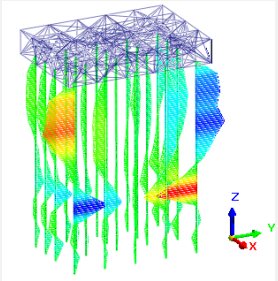


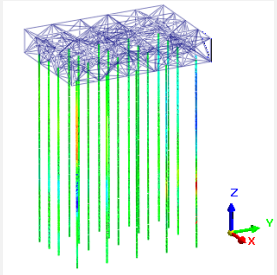










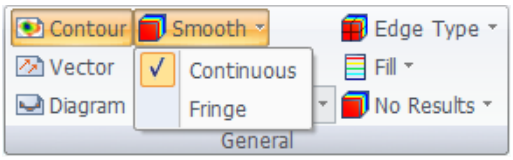


- Division
  Determine the number of divisions there will be on the diagram for representation.

2.4 Smooth

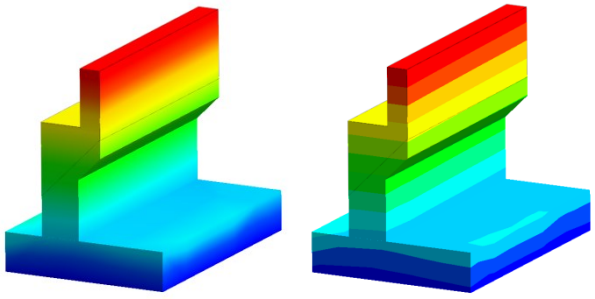
Overview

Display the analysis results using a continuous or fringe contour.



Details

▶Continuous
▶▶Fringe



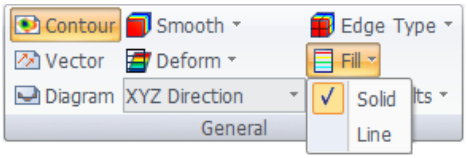
Continuous
Display the contours of the contour plot smoothly.

Fringe
Display the contours of the contour plot using bands for distinction.

2.5 Fill

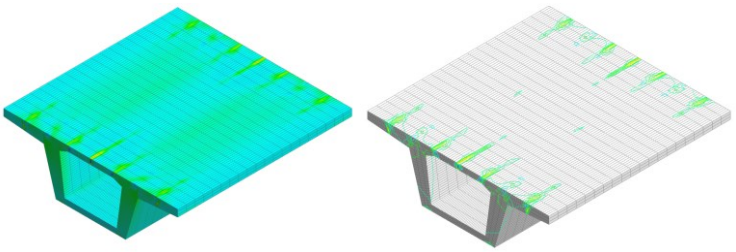
Overview

Determine the contour fill method. The available settings are Draw solid and Draw line.



Details

▶ Draw solid
▶▶ Draw line



Draw solid

Display the analysis result contour continuously on a solid.

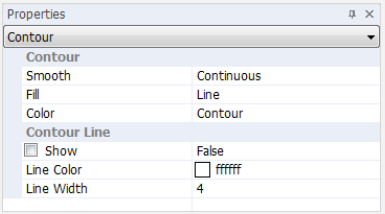
Draw line

Display the analysis result contour using isogram lines.

Tip

*** Adjust draw line width**

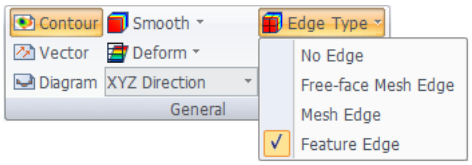
When selecting the Draw line fill, the thickness of the isograms lines can be changed on the contour settings of the Properties window.
Adjust the line thickness using Properties > Contour > Line Width.



2.6 Edge Type

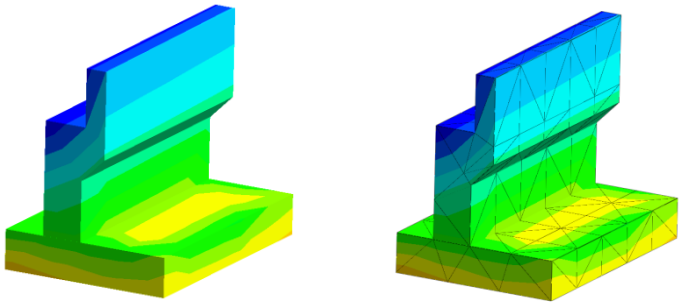
Overview

Define the mesh set display type of an activated analysis result graphic element.

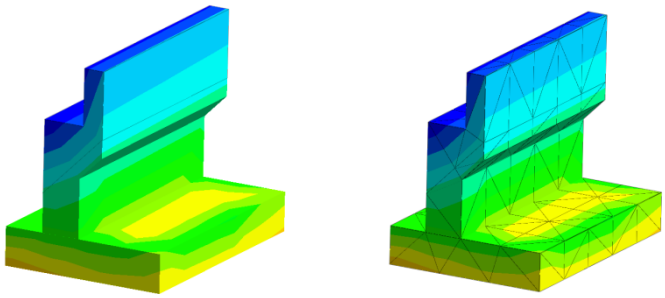


Details

▶No Edge
▶▶Mesh Edge



▶Feature Edge
▶▶Free-Face Mesh Edge



No Edge
The mesh is not drawn.

Mesh Edge
Draw the mesh edge.

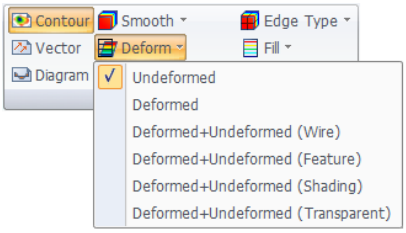
Feature Edge
Draw the feature edge of the mesh.

Free-Face Mesh Edge
Draw the mesh as a wireframe.

2.7 Deform

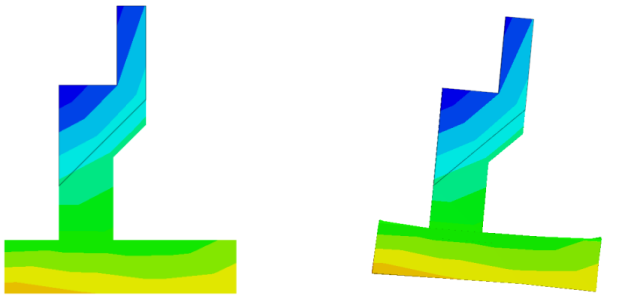
Overview

Determine the graphic processing of graphically represented results, with reference to the selected deformed shape.

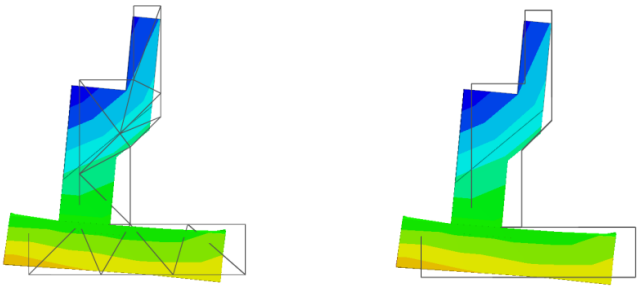


Details

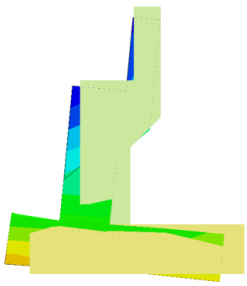
▶Undeformed shape
▶▶Deformed shape



▶Deformed + Undeformed (Wire)
▶▶Deformed + Undeformed (Feature)



▶Deform + Undeformed (Shading)



Undeformed
The deformed shape is not drawn. In other words, the original shape before deformation is graphically expressed.

Deformed
Output the deformed shape.

Deformed + Undeformed (Wire)
Output both the deformed shape and un-deformed shape (as a Wire).

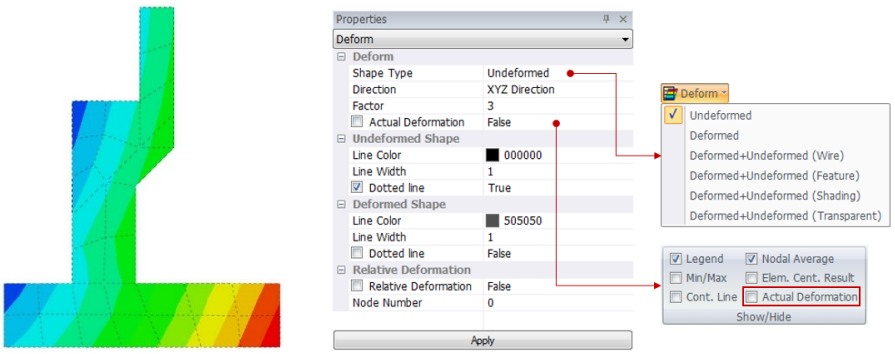
Deformed + Undeformed (Feature)
Output both the deformed shape and un-deformed shape (as a Feature boundary line).

Deformed + Undeformed (Shading)
Output both the deformed shape and un-deformed shape (using the geometry shape colors).

Deformed + Undeformed (Transparent)
Output both the deformed shape and un-deformed shape (using the geometry shape colors under the transparency condition).

Deform Properties
Specify the detailed properties for graphic processing of deformed shapes.

▶ Undeformed (Dotted Line)



[Deform]

- Shape Type
  Graphically display the results with reference to the selected deformed shape.
- Factor
  Input the scale value used when displaying the deformed shape on the screen.
- Actual Deformation
  Checked: Express the actual deformation.
  UnChecked: The program scales the displacement using an arbitrary scale to easily determine the deformed shape (Scale the maximum displacement to 1/20 size of the entire model).

[Undeformed/Deformed shape]
Specify the method used to display both the deformed shape and un-deformed shape simultaneously on the screen. It is used when applying Deformed + Undeformed (Wire) or Deformed + Undeformed (Feature).

- Line Color
  Specify the color of the line used to display the un-deformed/deformed shape.
- Line Width
  Specify the thickness of the line used to express the un-deformed/deformed shape.
- Dotted Line
  Switches to dotted type of the line used to display the un-deformed/deformed shape.

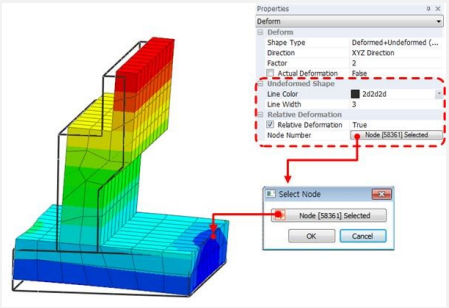
[Relative Deformation]

- Relative Deformation
  Express the node selected from the node number as a relative deformation.
- Node number
  Select the reference point of the relative deformation.

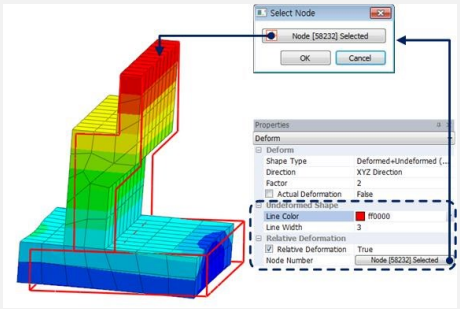
Tip

*** Relative deformation usage**

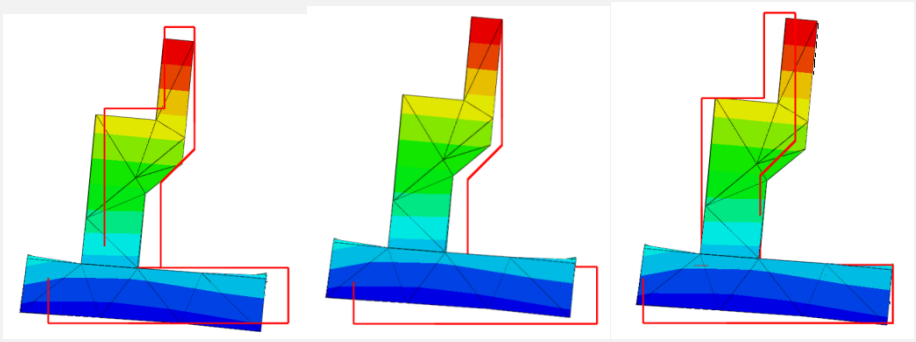
The standard for relative displacement examination can be selected to examine the deformed shape.



<Relative deformation based on bottom>



<Relative deformation based on top>

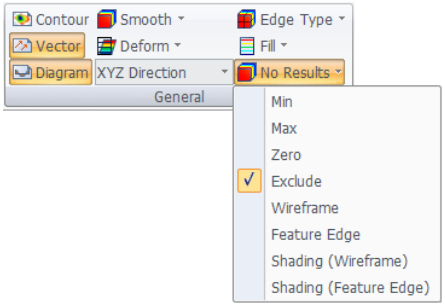


<Original> <based on Node 5232> <based on Node 58361>

2.8 No Results

Overview

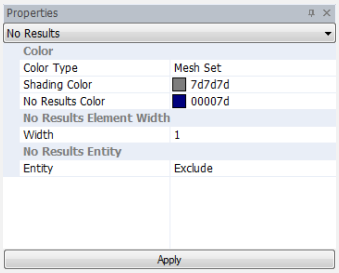
Determine the representation method for a target object with no results.



Tip

*** Property setting for no results**

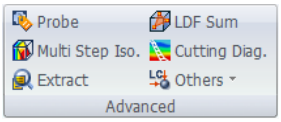
Specify the display settings for no result entities.



- Color Type
  Determine whether to display the specified color of the no result entity as a mesh set or user defined. When using the user defined method, the set shading color or no result color is displayed on the screen, depending on the display mode.
- No Result Element Width
  When displaying the no result entity as a wireframe, specify the wireframe width.
- No Result Entity
  Minimum value: Display a no result entity as the minimum result value in the contour process.
  Maximum value: Display a no result entity as the maximum result value in the contour process.
  Zero value: Display a no result entity as 0 in the contour process.
  Exclude: Exclude a no result entity from the contour process.
  Wireframe: Display the no result entity as a wireframe on the screen during the contour process.
  Feature edge: Display the no result entity as a feature edge on the screen during the contour process.
  Shading (Wireframe): Display the no result entity as a ‘shading’ that includes a wireframe on the screen during the contour process.
  Shading (Feature edge): Display the no result entity as a ‘shading’ that includes a feature edge on the screen during the contour process.

Section 3 Advanced Function

Various result outcome forms are provided for the results of a particular node or an element, such as result checking, extraction of result values, diagram etc. For data extraction, a MS-EXCEL compatible result table is provided to ease the result check for each stage.

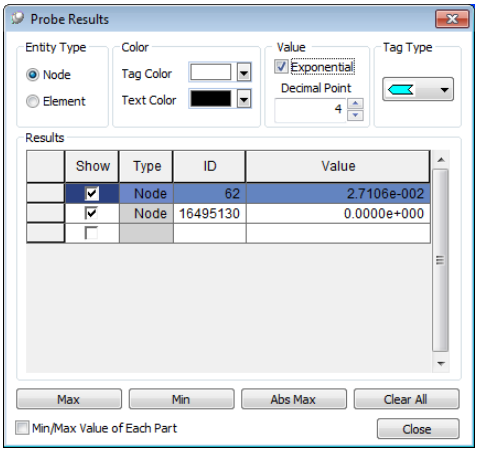


3.1 Probe

▶Probe result

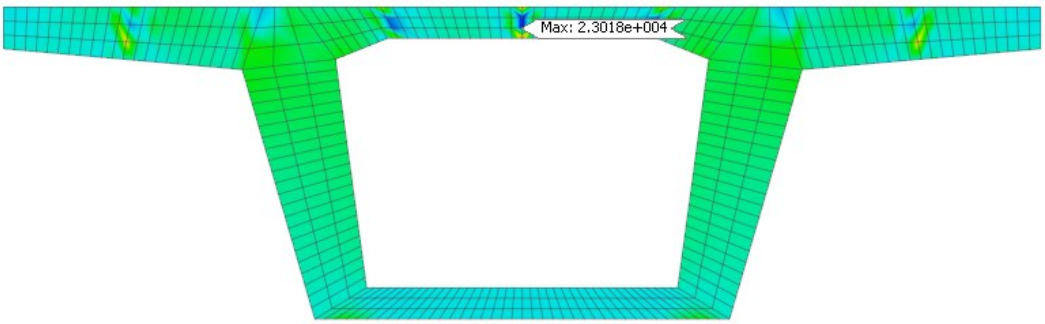
Overview

Display the results by attaching a result tag on the desired node or element.



Methodology

Select the node or element on the screen to create a probe. The user can edit the tag color, text color, tag type etc.
The information and values of a node/element, where the maximum, minimum, maximum absolute values are generated, on the current result can be checked, and the minimum, maximum values of each mesh set can be checked.

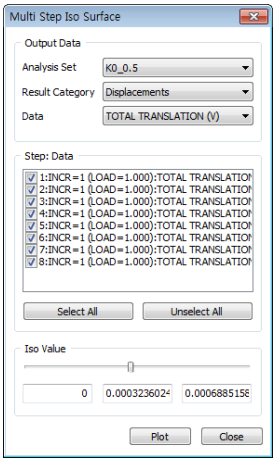


3.2 Multi Step Iso Surface

▶ Multi step iso surface

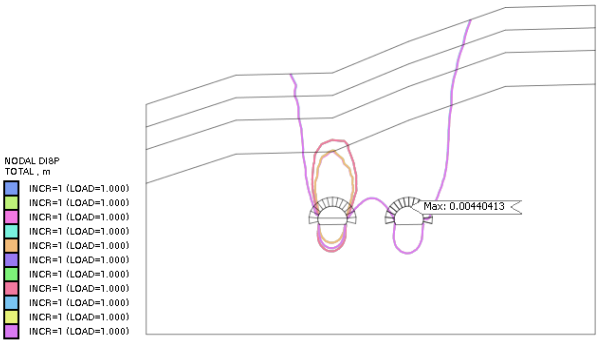
Overview

Simultaneously display the iso-surfaces obtained from multiple analysis steps.



Methodology

Select the analysis condition to display the iso-surface on and specify the result type, result value and step. Input the iso-value. The iso-value can be selected using the bar between the maximum/minimum values of the total result.

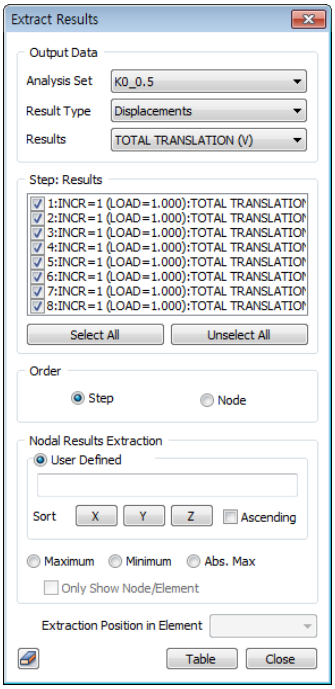


3.3 Extract

▶Extract Results

Overview

Extract the user desired data from the analysis result.



Methodology

Select the analysis conditions and result type of the extraction target, and specify the results.
The result types differ, depending on the analysis model setting (2D/3D) and material property (Plain strain, Plate, Solid etc.).

[Step: Results]
Specify the result extraction target step from the results. To select all steps in the list, click the [Select All] button. To deselect all selected steps, click the [Unselect All] button.

The Order are [Step] or [Node].
For [Step], the extracted result values are listed on a table in the order of affiliated step. For [Node], the extracted result values are listed on a table in the order of node coordinates for nodes, and the center of mass coordinates for elements.

The nodes, from which the result values will be extracted, can be selected by directly entering the node ID number or selecting the nodes on the screen. It is possible to extract only the maximum, minimum and absolute minimum values. Checking [Only Show Node/Element] extracts the maximum/minimum value within a node/element displayed on the screen.

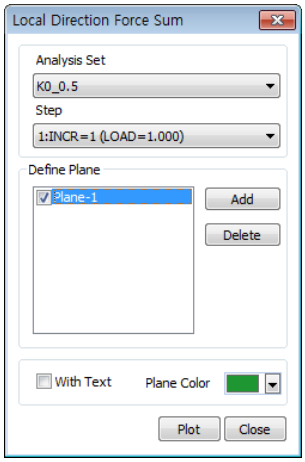
The element extraction location, where the result values will be extracted, can be directly specified for a 1D element.

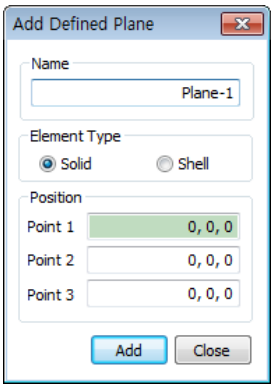
3.4 Local Direction Force Sum

▶Local direction force sum

Overview

Calculate the force sum on an arbitrary section of a Solid or Plate element.





Methodology

Select the analysis case and analysis step, and create the section plane to calculate the local direction member force sum.
Check [With text] and click the [Plot] button to output the section force component at the calculated centroid of the section as a text file, and to display the define plane on the screen.

Tip

The Local Direction Force Sum is a useful function that can automatically calculate the section force of a Solid or Plate element. This function calculates the internal force sum in each direction at the centroid of an arbitrary plane on a Solid or Plate element, using the internal force of each node on the plane. The Local Direction Force Sum is useful when computing the member forces to be applied in structural design, after detailed analysis on a particular section of the structure.

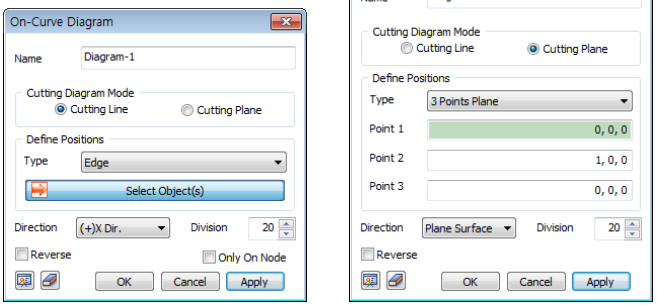
The arbitrary section is determined from the intersection between the structure, composed of a Plate or Solid element, and the infinite plane created by selecting 3 arbitrary points. The centroid is automatically calculated for this determined section and the section force, in the same direction as the frame result, is calculated using the stress results of that analysis step.
If a structure that is not a considered exists in the specified plane, the mesh set corresponding to that structure needs to be deactivated before specifying the plane.

3.5 Cutting Diagram

▶ Cutting diagram

Overview

Specify an arbitrary line or plane to check the on-surface analysis results as a diagram.

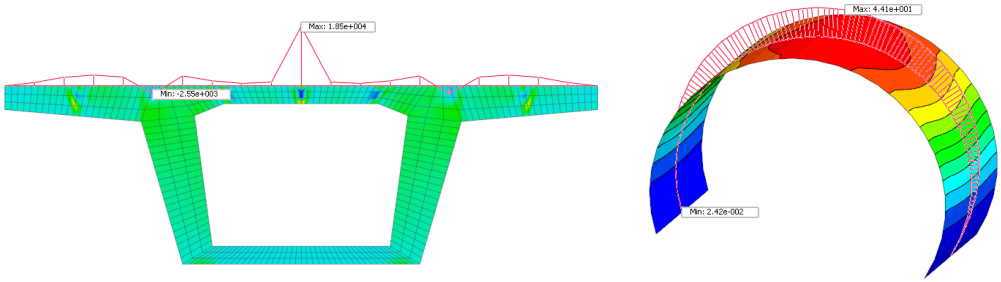


Methodology

The diagram setting method can be specified as [Line] or [Surface], and the results can be displayed as a diagram.

[Cutting Line]: Select a 2-point line or edge when defining the position.
For the 2-point line, select the coordinates directly on the screen or input the coordinates of the 2 points.
Then, specify the direction of the diagram and input the number of samples to divide the diagram by that number and display on the screen.
The diagram direction follows the GCS and the default direction is the (+) direction. To draw the diagram in the opposite direction, check the [Reverse] option. For the edge, select [Only On Node] to draw the diagram on that node position. Only one edge can be selected.
The arbitrary line diagram is useful when assessing the relative settlement conditions of a geotechnical structure for 2D models.

[Cutting Plane]: Select a 3-point face or plane when defining the position.
For the 3-point face, select the coordinates directly on the screen or input the coordinates of the three points.
The diagram direction can be selected in the [Plane Surface] or [Plane Normal] direction, and the [Reverse] option can be checked to draw the diagram in the opposite direction.
The sample number is the number of diagram divisions displayed on the screen.
The arbitrary plane diagram is useful when checking the member force distribution in a particular section of a 3D model (ex. Lining transitional section model).



Tip

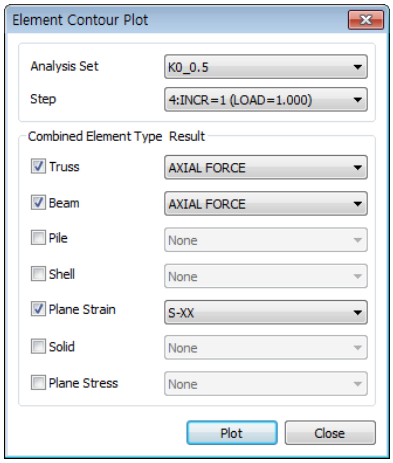
The specified arbitrary line diagram is registered on the left Works tree > Result tab and the check button can be used to show/hide on the screen. The right mouse click can be used for editing.

3.6 Others-Element Contour Plot

▶Element contour plot

Overview

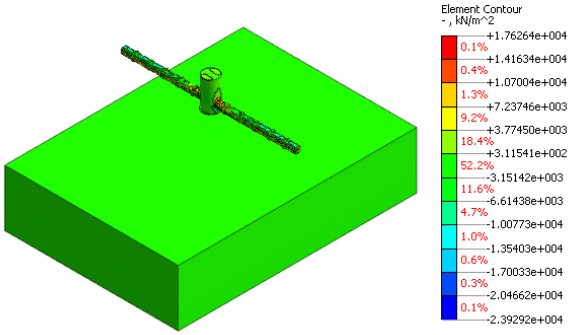
Print different element result types on the screen.



Methodology

Specify the result set and step, and select the element type and element results to print.

▶Simultaneous representation of ground stress and plate stress

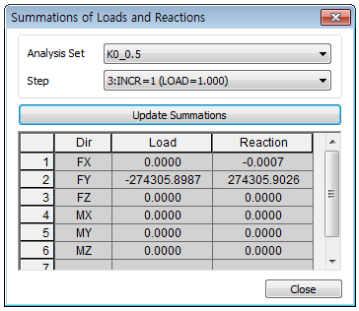


3.7 Others-Summation of Reaction

Overview

Display the reaction force sum on a table. The table can be checked for all analysis that has the reaction forces as its result output.

▶Load/Reaction force sum



Methodology

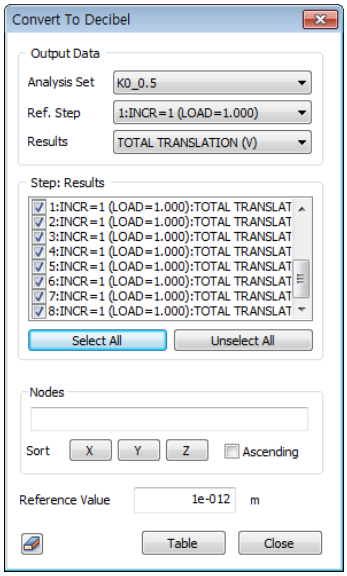
Specify the analyzed analysis case and step, and click Update Load sum/Reaction force sum to automatically calculate and display the Load/Reaction force sums on a table.

3.8 Others-Convert to Decibel

Overview

Convert the displacement, velocity and acceleration to Decibel (dB) form and display it as a table.

▶Decibel conversion



Methodology

Select the node after specifying the analysis set/step/result and input the reference value to convert the displacement, velocity and acceleration at that node to Decibels (dB).

$$N(dB) = 20\,log_{10}\left(\frac{X}{X_{ref}}\right)$$

Tip

Vibration levels generally use g (acceleration) or g^2 values, while the dB (decibel) is widely used. This is because when noise or vibration occurs, the physical quantity range humans can feel is very wide and so, it is inappropriate to express this physical quantity using a linear scale.
The decibel (dB) is expressed using a ratio between the physical quantity X and the reference physical quantity X_{ref}, plotted as a common log function.
Here, when the reference physical quantity is 20μPa for sound pressure, the acceleration is in μm/s2 (KS, JIS). The 20μPa is a subjective reference that is expressed as "the minimum sound pressure an average healthy human can hear"

[Decibel Reference Levels(ISO R 1683)]

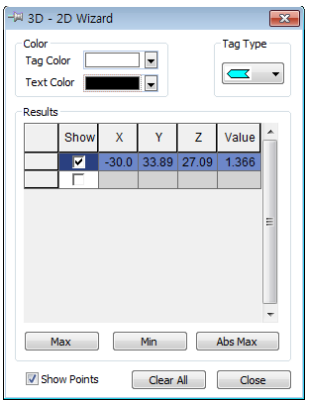
	Quantity	Definition	Ref. level
Amplitude Rations	Vibratory Acc. Level	$L_a = 20 \log_{10}(a/a_0)$ dB	$a_0 = 10^{-6}$ m/s^2
	Vibratory Vel. Level	$L_v = 20 \log_{10}(v/v_0)$ dB	$v_0 = 10^{-9}$ m/s
	Vibratory Dis. level	$L_F = 20 \log_{10}(d/d_0)$ dB	$d_0 = 10^{-12}$ m

3.9 Others- 3D-2D Wizard

Overview

When checking the results after 3D analysis, use the cut section & clipping function to check the arbitrary section result contours. Here, for the cut arbitrary section, the analysis results such as the minimum, maximum, absolute maximum value etc. can be checked along with the position. Because all results from the FEM analysis is calculated with reference to the element nodes, when a cut arbitrary section passes through the interior of the solid element, the results from nearby nodes are automatically interpolated and output.

▶3D-2D Wizard
(Result output on arbitrary section)

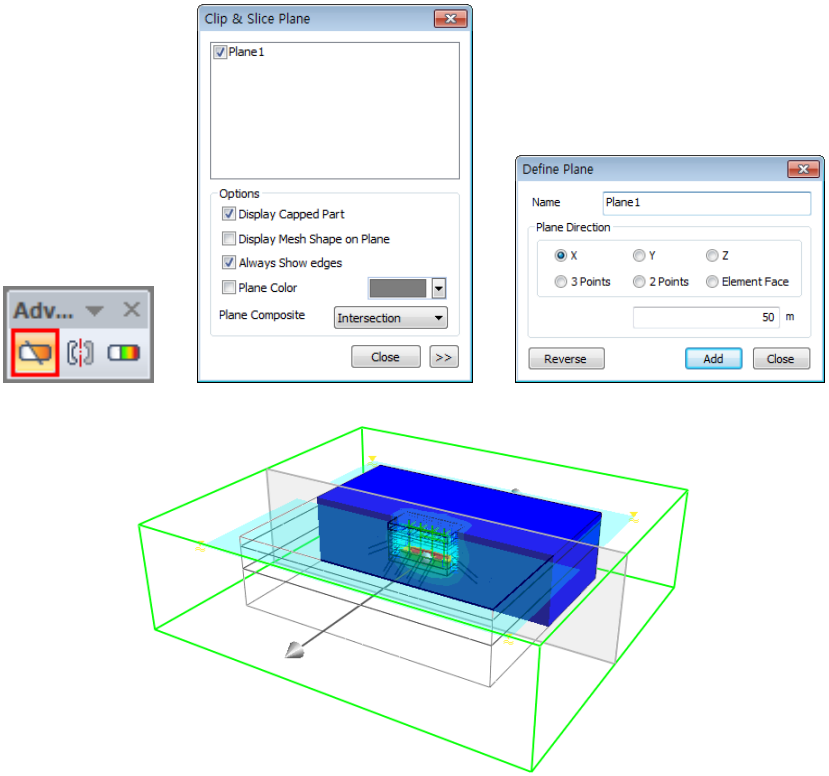


Methodology

The result tag from the specified arbitrary section is output following the process outlined below.

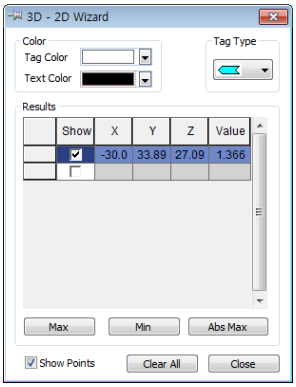
1. Add the arbitrary section where the results will be checked, from Additional view control toolset > View cut model (Cut section & Clipping). As many cut sections as desired can be added in the GCS axis direction, or in the arbitrary section direction.

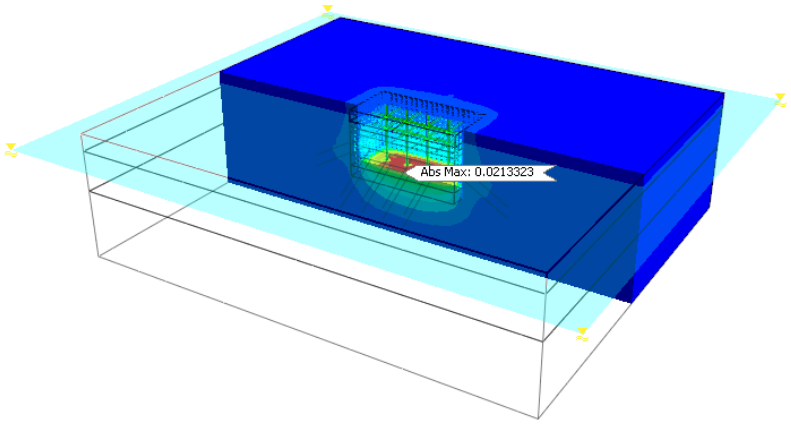
▶Cut section & Clipping
(Specify arbitrary section)



2. Choose Result analysis > Advanced > Others > 3D-2D Wizard.

Check the [Show Points] option to display all nodes on an arbitrary section where the results can be output. Like the result value tag, select the nodes or elements to check the numerical value, along with the positions of the minimum, maximum, absolute maximum values on the viewed section.

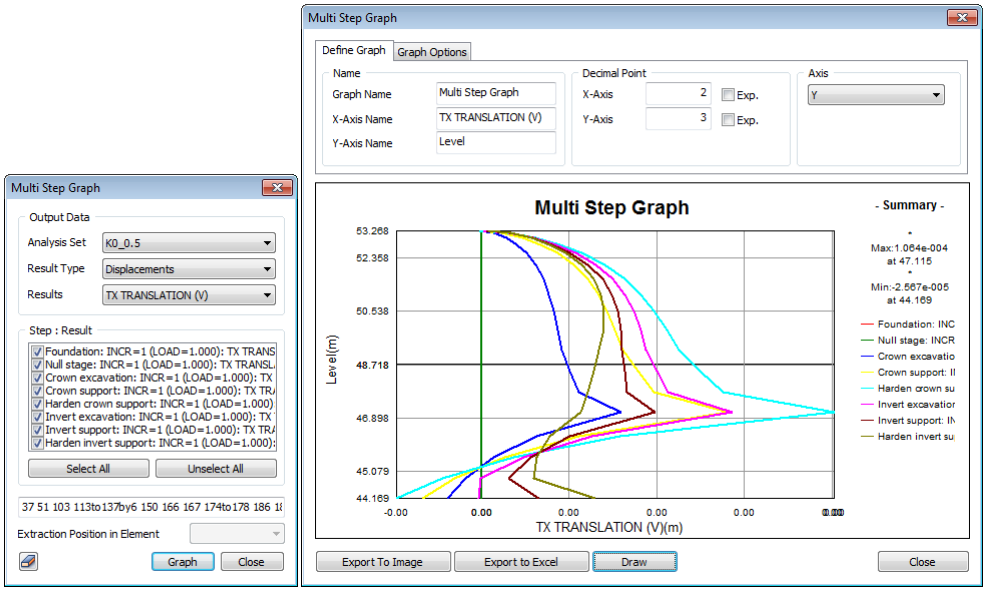




3.10 Others- Multi Step Graph

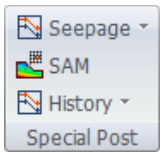
Overview

The results of multi-step are drawn by graph type based on the selected nodes/elements. Analysis set, result type, results, step and nodes/elements will be selected to draw graph. In the 'Define Graph', 'Axis' is the coordinate of selected nodes/elements, and it is placed at the Y axis of graph. The value of selected nodes/elements is placed at the X axis of graph.



Section 4 Special Post

Seepage analysis result (Flow Quantity/ Flow Path), Slope stability analysis (SAM) result, Time variant hysteresis graphs can be identified.

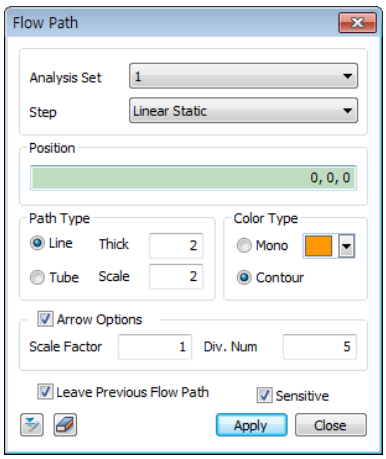


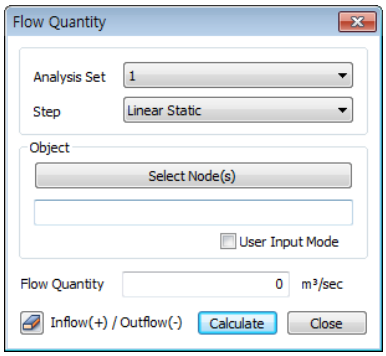
4.1 Seepage

▶Flow Path
▶▶Flow Quantity

Overview

Identify the flow path and flow quantity values obtained from the Seepage analysis results.





Flow Path

Methodology

Specify the analyzed analysis case and step, and then input the coordinates of the flow path display position (an arbitrary point on the flow path). Snap option can be used to input the coordinate values.

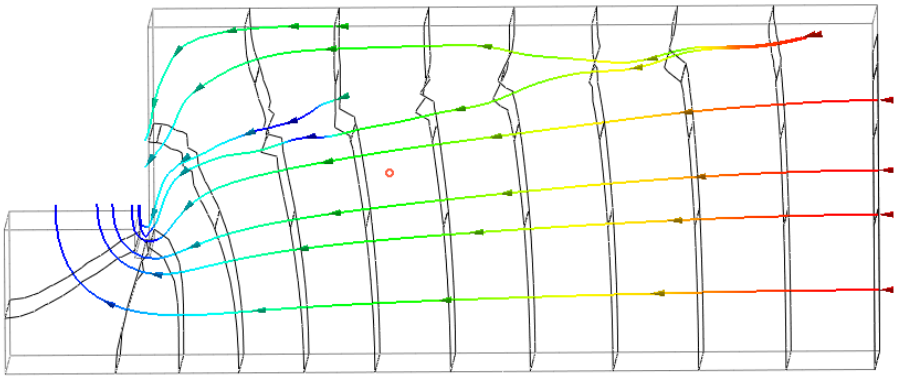
The flow path can be displayed as a line or pipe. For a line, the line thickness and for a pipe, Scale can be used to adjust the size.

The flow path color can be selected between Single color and Contour.

The arrow grid option is an indicator that represents the flow direction of the flow path. The scale value and number can be used to adjust the arrow size and number of arrows.

If the [Leave Previous Flow Path] option is checked, a flow path is displayed for every clicked position on the screen. If this option is unchecked, a flow path is displayed only on the last clicked position.
If the [Sensitive] option is checked, the flow net is displayed on the screen immediately. If this option is unchecked, the [Apply] button needs to be pressed in order to display the flow net.

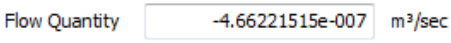
▶Flow path



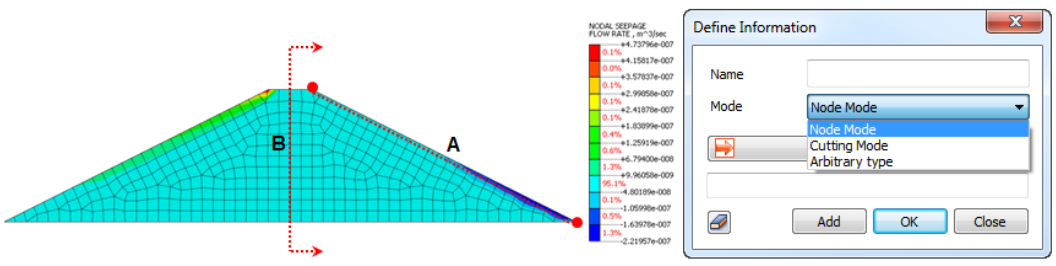
Flow Quantity

Methodology

Specify the analyzed analysis case and step, and then select the nodes where the flow quantity will be calculated.
The nodes can be selected and specified on the screen, or [User Input Mode] can be checked to input the node ID if the node number is known.
Press the Calculate button to automatically calculate the flow quantity at the nodes, and the calculated flow quantity value is expressed on the dialog.

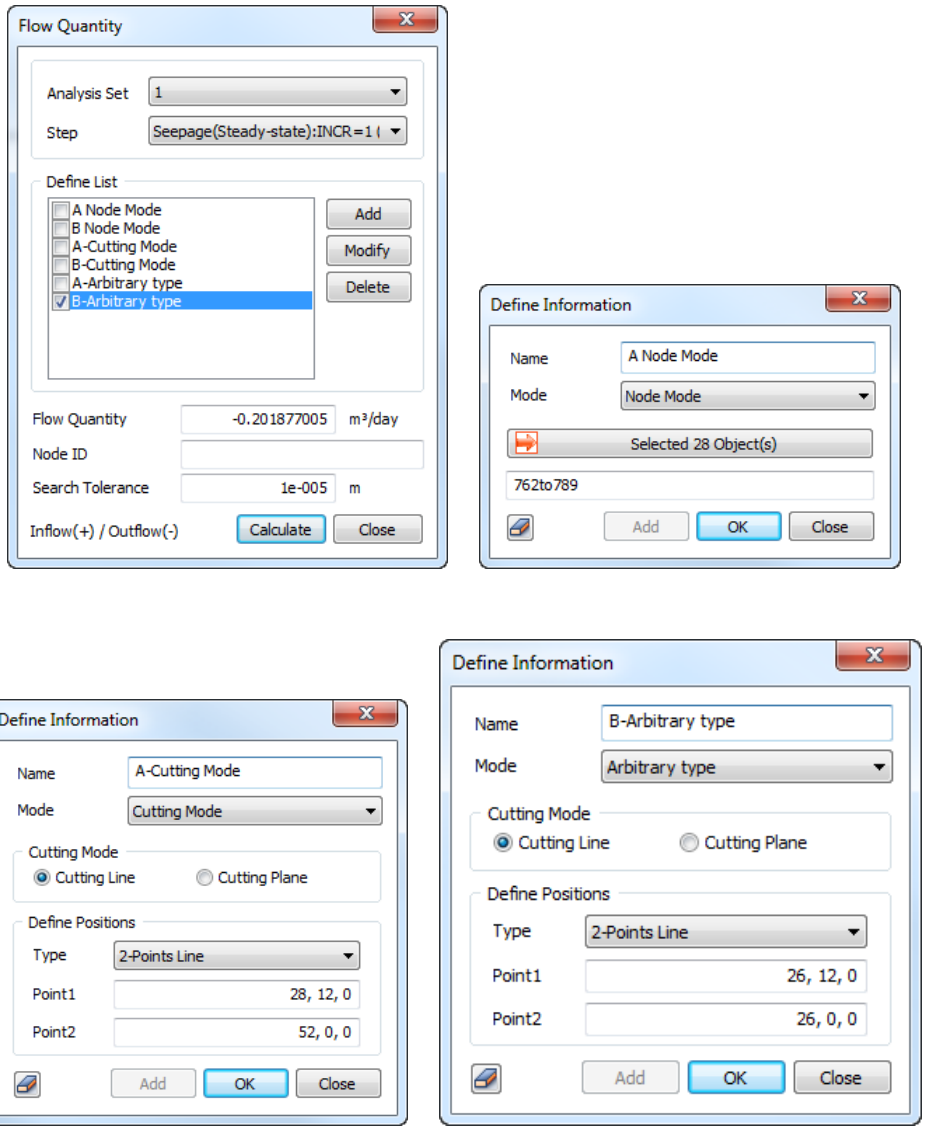


The flow quantity at each node is made up of the flow rate value sum, and (+) indicates inflow while (-) indicates outflow.
The node selection method (Node Mode), line/plane definition method (Cutting Mode) and Arbitrary Type are supported.
Nodes/Cutting Mode - returns the flow rate by summing the flow rates calculated at the nodes by direct selection or input of the nodal points where the outflow or inflow occurs.
Arbitrary type - calculates the flow of elements through any line or plane.



There are ‘Cutting Line’ and ‘Cutting Plane‘ option in the ‘Cutting Mode’. The ‘Cutting Line’ option calculates flow quantity from nodes within the 'Search Tolerance' by defining ‘2-Points Line’ directly or selecting ‘Edge’. The ‘Cutting Plane‘ option calculates flow quantity from nodes within the 'Search Tolerance' by defining ‘3-Points Plane’ directly or selecting ‘Plane’. The ‘3-Points Plane’ type calculates flow quantity from the infinite plane consisting of three points or the plane only consisting of them with the 'Limited Plane' option.

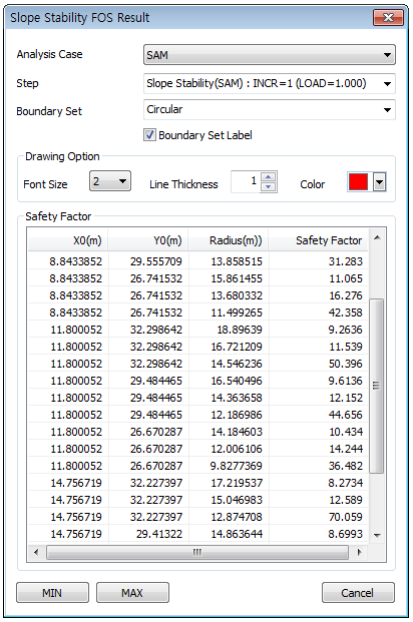
Once the information for calculating flow quantity is created in the ‘Define Information’ dialogue box, it will be registered in the ‘Define List’. Flow quantity can be calculated by duplicate selecting a number of information registered in the ‘Define List’ and they can be modified or deleted as well. The node information included in the checked ‘Define List’ is displayed in the ‘Node ID’ and duplicated same nodes of a number of information will be treated as one.



4.2 Slope Stability (SAM) Result

Overview

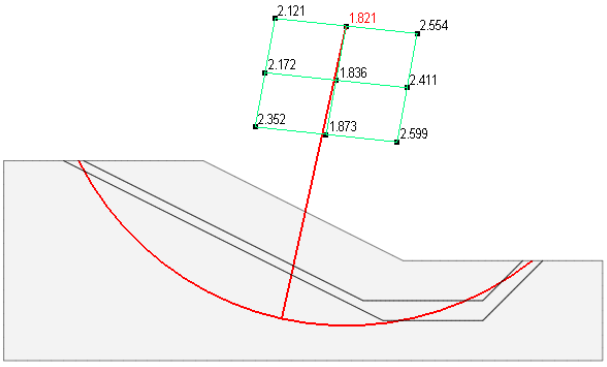
Output the Slope stability analysis results that incorporate the limit equilibrium method indicators.



Methodology

Specify the analyzed analysis case and analysis step to check the results on the virtual fracture plane defined in the stage. Check [Boundary Set Label] and specify the Boundary set to identify the boundary conditions of the virtual fracture plane specified on the screen.
Use Min, Max to check the area where the maximum/minimum values occur, and use the Drawing option to modify the thickness, safety factor, size and color of the fracture plane at that point.

▶SAM result

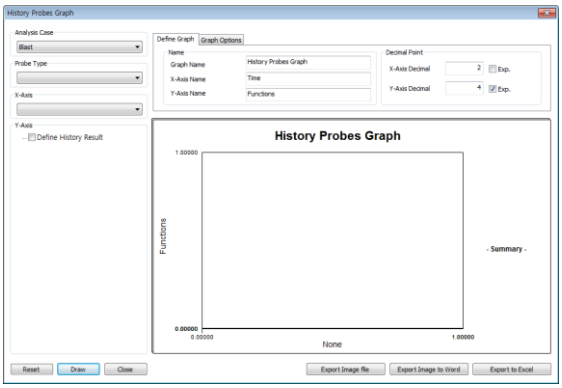


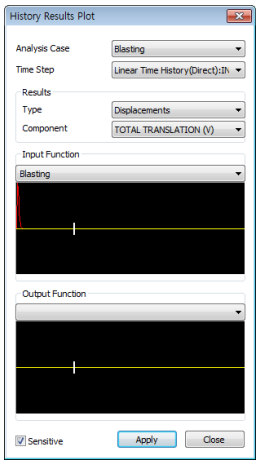
4.3 History

Overview

Identify the results at particular positions for a time-existing analysis case, using a time variant graph or plot the results for each time period on the work screen.

▶History result





Graph

Methodology

Select the analysis case for result output and function type for each analysis type. The function type is possible for analysis cases with time (Transient seepage, Consolidation, Fully coupled stress-seepage, Linear/nonlinear time history analysis, 2D equivalent linear).
The probe type can select the specified results from Analysis > Special Post > History, and the results for each probe type are as follows.
※Be aware not to include a search type that is not output in the analysis case.

Probe type	Result type
Displacement/Velocity/Acceleration	Displacement, Velocity, Acceleration, Relative displacement, Relative velocity, Relative acceleration
Truss/ Embedded truss/ Geo-grid(1D)	Strain, Stress, Member force, Seepage, Hinge Force, Hinge Deform
Beam	Strain, Stress, Member force, Seepage, Hinge Force, Hinge Deform
Plane strain	Strain, Stress, Member force, Seepage
Plane stress/Geo-grid (2D)	Strain, Stress, Member force, Seepage
Axis symmetric	Strain, Stress, Seepage
Shell	Strain, Stress, Member force, Seepage
Solid	Strain, Stress, Seepage
Response spectrum	Relative displacement, Relative velocity, Relative pseudo-velocity, Absolute acceleration, Absolute pseudo-velocity
Transfer function	Displacement, Velocity, Acceleration
Seepage node results	Total head, Pressure head, Flow rate
Point Spring/Elastic Link	Strain, Stress,Member Force, Hinge Force, Hinge Deform

The output function used on the vertical axis of the result graph is selected on the Y axis. This function must be defined in advance using Analysis > Search time history result.

The name of the graph, X axis and Y axis of the time history graph can be defined using Define graph, and the values can be expressed in exponential form.
The axis settings for the X axis and Y axis can be set in detail using the Graph option. The max/min value can be determined, and line shape, line width, line form can be set. It can also be expressed using a log scale.

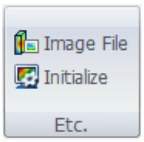
The generated graph is displayed in the dialog window, and can be output in various forms using Print image file, Print image word file and Print excel file.

Plot

Methodology

Select the analysis case for result check and specify the time step. The number of created time steps is determined from the number of total time steps defined in Analysis > Analysis setting.
Selecting the analysis result output type and results automatically displays a contour on the screen.
The time history function reflected in the analysis case can be checked in the [Input Function]. The specified time of the time step can be checked on the input function, and the output results reflect the selected time on the time step.
The [Output Function] can be specified in Search time history analysis, and the output result function can be checked. The specified time of the time step can be checked on the input function, and the output results reflect the selected time on the time step.
Check the [Sensitive] option to immediately output the results from the selected time step on the screen.

Section 5 Etc.

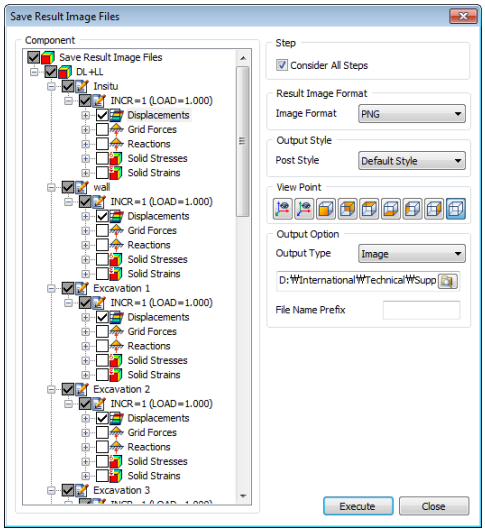


5.1 Image File

▶Image file

Overview

Output the finished analysis results as an image file, or a word file that contains an image file.

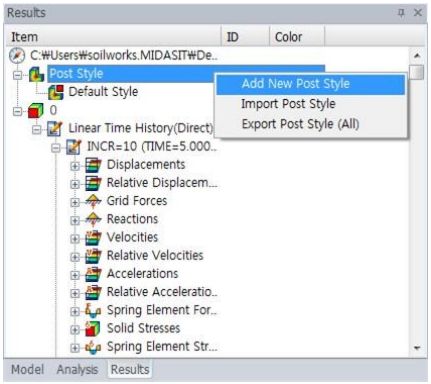


Methodology

Check the desired article on the component tree menu and determine the image save format.

[Consider All Steps]: Function which allows to Select/Unselect Stage/Step/Result/Component level at the same time. If this option is unchecked, it will Select/Unselect individually.
All the subordinate components are checked for all stages when the result level is selected.
Only components are checked for all stages when the component level is selected.

FEA NX supports PNG, JPG, BMP image file formats.
Select the Post style. The Post style can be used to conveniently identify different result data, after the current post-processing graphic display options are saved on the Result tree.
Call up the Context menu on the left Works tree by right mouse click, and use the [Add New Post Style] menu to save the current graphic display settings. Double click the saved post-processing style to apply that post style.



Post styles from a different model file can be imported by using the [Import Post Style] on the Context menu. Also, the [Export Post Style (All)] menu can be used to save the set Post style on the current model file separately.
The View point is provided to easily determine the model direction when capturing the screen.
The results can be saved as an image file or a word file using the Output form, and specifying the output location saves the image or word file on that location.
The File name prefix is the prefix that is automatically assigned in front of the file name during save.

5.2 Initialize

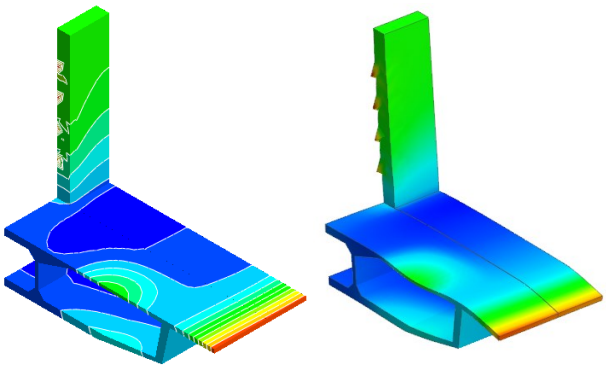
Overview

Reset the post to the basic post-processing condition set by the program.

Methodology

Clicking Reset post resets all post-processed results to the basic post-processing condition. The default Initial value is a Continuous contour, the Result edge a Feature edge and the Deformed shape an Underformed shape.

▶Before initialized
▶▶Initialized

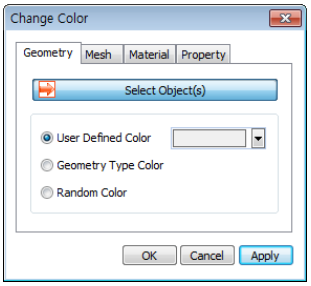


Section 6 Geometry

6.1 Change Color

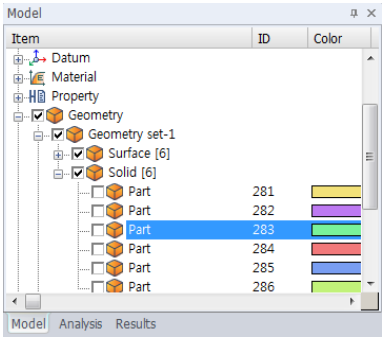
Overview

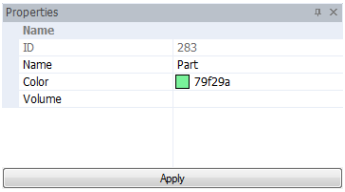
Edit the color of a Geometry/Mesh/Material/Property.



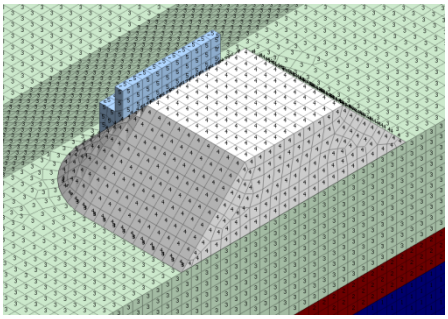
Methodology

Specify the color and object, and click the Apply button. The color of Geometry/Mesh/Material/Property can be changed easily by this function. Also the color can be edited by using property window.





[User Defined Color]: User can define the color of selected object.
[Geometry Type Color]: Change color to geometry type color.
[Random Color]: Apply random color to selected object.
[Display ID]: Check to see Material/Property ID.



6.2 Color Type

Overview

Display the face or solid geometry shape or mesh set shape in the shape color or material color on the screen.

Methodology

Select the shape color or material color.
This function can be used to check what types of material properties are assigned after modeling.

6.3 Random Color

Overview

Assign a random color to a Face/Solid geometry shape or Mesh set.

Methodology

The program automatically assigns a color to the geometry shape or Mesh set.

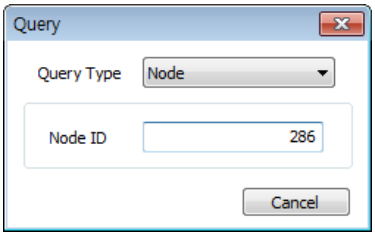
A random color can be assigned to a geometry shape or Mesh set by clicking the Geometry shape () or Mesh set () on the [View toolbar] on the top of the work screen.

Section 7 Node/Element

7.1 Query

Overview

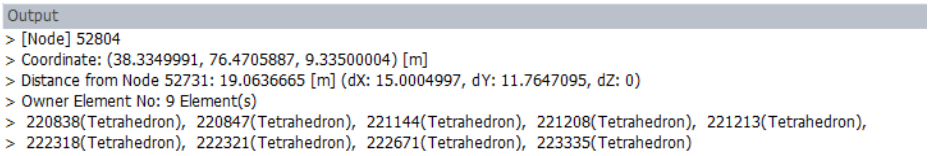
Check the Node or Element information.



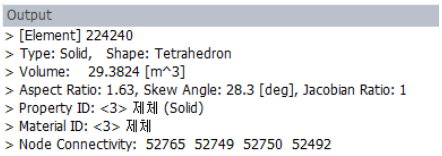
Methodology

Specify the Query type (Node or Element) of the target and input the Node number or Element number. The particular Node or Element can be directly selected on the work screen. If the Query type is a Node, its coordinates and adjacent elements to that node are printed on the Output window. If the Query type is an Element, the type and shape, volume, aspect ratio, property/material and node connection information are printed.

▶Node information



▶Element information



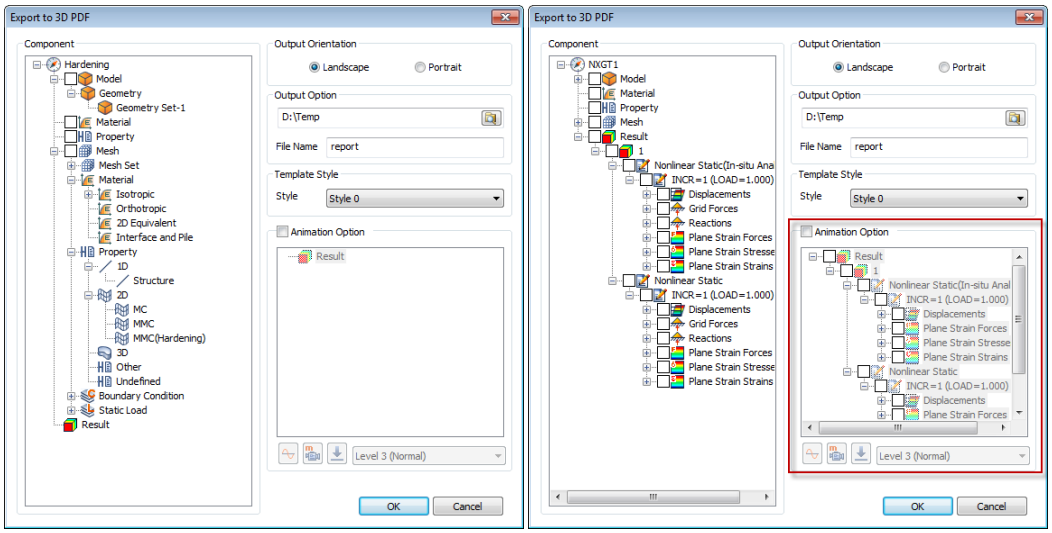
Section 8 Export 3D PDF

8.1 Export to 3D PDF

Overview

Save the model information before analysis and the result information after analysis as a 3-dimensional PDF document. Adjusting the 3D view on the PDF document and checking the cross-section information, similar to the operations in the program, is possible. Hence, the 3D model/analysis result information can all be checked on one PDF document without the model/result files.

▶Before analysis (Model information)
▶▶After analysis (Result information)
▶3D PDF



Methodology

Export all the information on a model such as table for input parameters, geometry and mesh before analysis as a PDF file. Specify the output direction, path, file name and select the OK button to create a PDF document as shown above.
Because this operation has model tree/view tools etc. similar to the program, basic show/hide, rotation, move operations are available and can be used to check the 3D model information.
In particular, the model dimension lines and comments can be added and for 3D models, the arbitrary cross-section information can also be checked.

After analysis, the result images and animation can be saved and the output results can be selected on the tree structure and composed as a PDF document.

Section 9 Flight Simulation

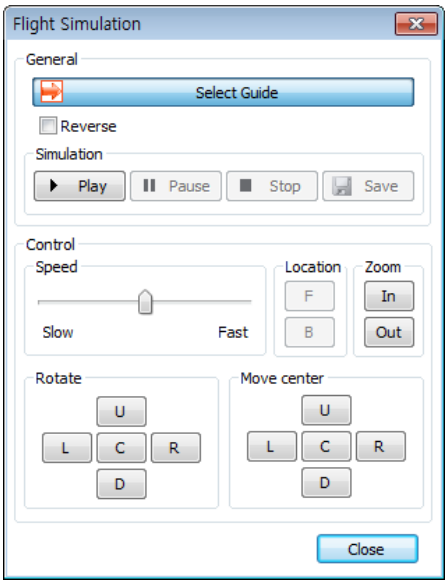
9.1 Flight Simulation

Overview

The Perspective view function from the View toolset below can be used to check the interior information and results of a 3D model.

For the Perspective view function, the checking positions need to be checked directly using a mouse, but to automatically simulate it along a particular guideline, the 'Flight simulation' function can be used.
The guideline can be selected from a line or repeating line(wire), and the model interior can be checked along the selected line. This function is available for both before/after analysis.

▶ Flight simulation



Methodology

Select the guideline. Directly draw the path in the model interior, or select a sub-shape line of an existing geometry shape.
Selecting the line instantly displays an arrow in the processing direction, and the direction can be changed using the opposite direction option.

The simulation operates using Start, Stop, End buttons and selecting the Record button saves the simulation as a video file.

[Control]
Control the process speed using a scroll bar.

In particular, the (Progress/Rewind) button is activated at the Stop state and the model interior results can be checked by moving back and forth along the guide line. The currently viewed model can also be zoomed in/out.

[Rotate]
The view perspective can be rotated Left/Right/Top/Bottom as it moves along the guideline and can be moved (offset) with respect to the guideline. Hence, the information can still be checked when the guideline position is different from the target position by using the rotate/move operations.

All buttons during the flight simulation can be used as the following shortcut keys, and can be checked from the Tooltips of each button.

Shortcut Key List	
A	View left
D	View right
S	View bottom
W	View top
1	Rotate to original position
←	Move center to the left
→	Move center to the right
↑	Move center to the top
↓	Move center to the bottom
2	Move center to original position
R	Proceed from stop state(Only available at Stop state)
F	Rewind from stop state(Only available at Stop state)