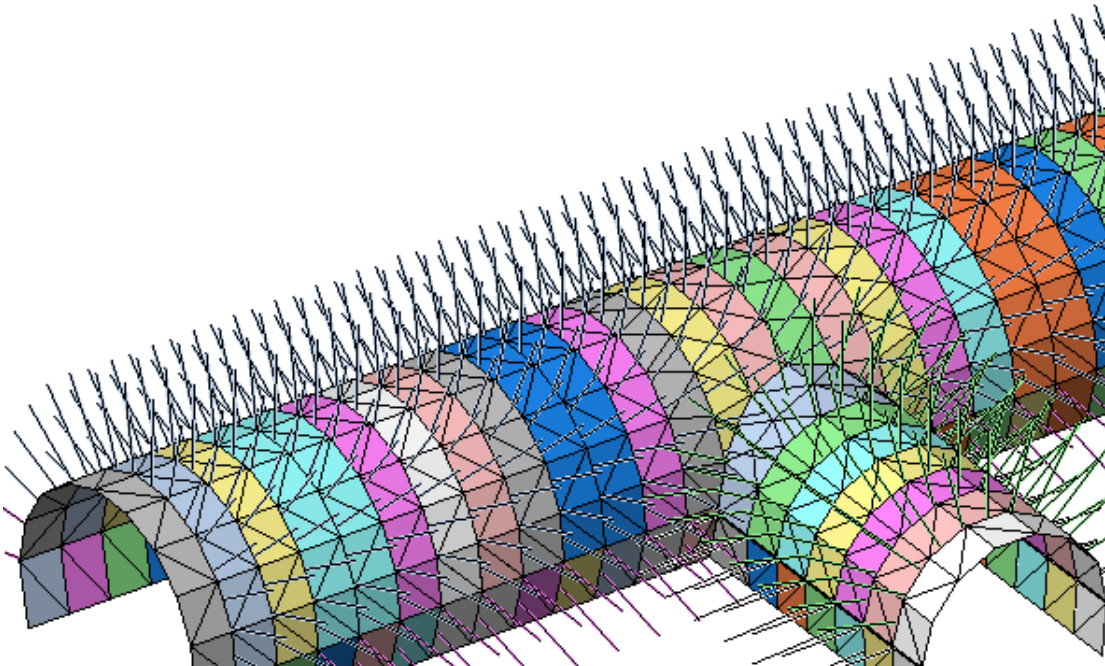
midas GTS

GTS V300 Release Notes

"Next Generation Solution in Geotechnical and Tunnel Engineering"



midas GTS

GTS V300 Release Notes

"Next Generation Solution in Geotechnical and Tunnel Engineering"

midas GTS

GTS V300 Release Notes

OVERVIEW

Documentation describes the modifications for midas GTS V300 R4.

I) Improvements

- Hoek-Brown Model
- Modified Mohr-Coulomb
- Cam Clay and Modified Cam Clay Models
- Tabular Input for Material Properties
- User Defined Phreatic Surfaces
- Input Data ASCII File
- Non-symmetric Multi-Frontal Direct Solver
- File Exchange with midas Civil and midas Gen
- Quadrilateral and Q-morph Mesher
- Align Edge Seed Near Inner Loop
- Mesh Clean-Up
- Post Remesh
- Align edge seed near inner loop
- Mesh Clean-Up
- Auto Mesh Solid : Advanced Option : Post Remesh
- Mesh Preference : Seeding Options
- Measuring Element Properties
- New Extruding Feature
- Creating a single node using Center of Mass Summation
- Create a surface using solid or 2D models.

II) Bug Fixes

- Analysis
- Pre/Post Processing

III) Known Issues

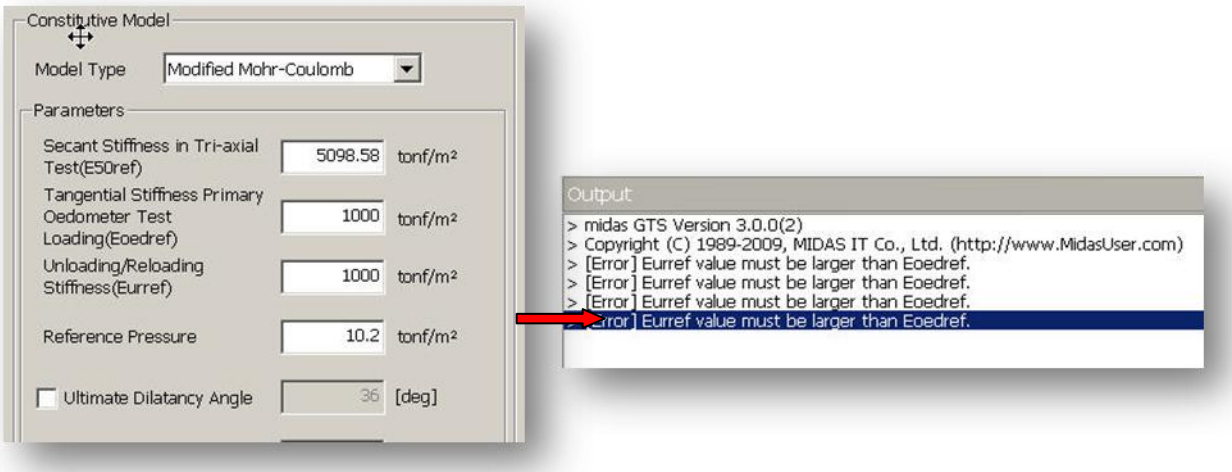
Hoek-Brown Model

In GTS V251, the return algorithm by Euler forward method uses an internal tolerance of 1.0 e-6. In GTS V300, instead of using the Euler forward method, a Cutting plane method has been implemented. This method is similar to the Euler Backward method, which uses an internal tolerance of 1.0e-3 and a routine that improves convergence by internal discretization of strength. In previous implementation, an Erroneous formula that was in the first order gradient equation has been corrected.

Modified Mohr Coulomb

a) In GTS V251, the Modified Mohr Coulomb (MMC) model has been updated such that the behavior at low cohesion values is more robust in this version. Further, in case initial stresses are defined for the MMC model elements, the size of the initial yield-surface is now automatically calibrated to the initial stresses in the respective integration-points, such that larger load-increments can be applied in initial loading of the elements.

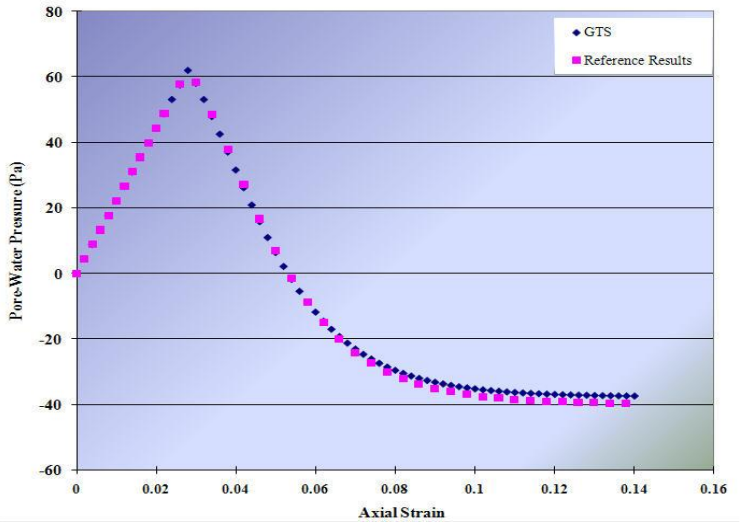
b) In the Modified-Mohr Coulomb material model, if the user inputs Unloading/Reloading Stiffness (E_{urref}) equal to or less than the primary loading stiffness (E_{oedref}), a warning message in the Output window is displayed suggesting the value should be larger.



Cam Clay & Modified Cam Clay

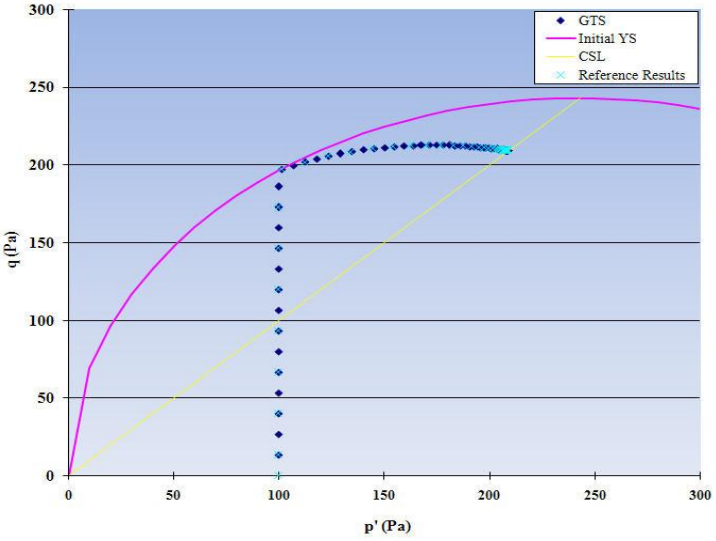
midas GTS can now represent undrained and drained behavior of soft clays and for-saturated and unsaturated conditions.

1) **Cam Clay model**:
Pore-Water Pressure (Pa) vs. Axial Strain for drained conditions



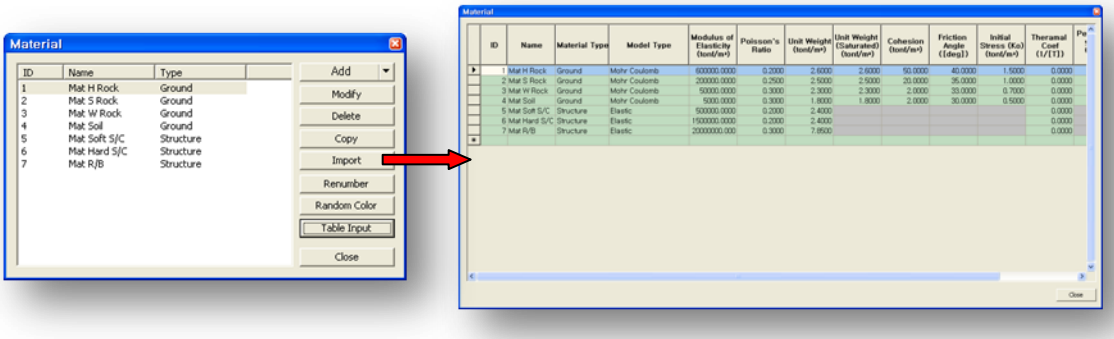
2) **Modified Cam Clay model**:

von Mises (Pa) vs. mean effective stress (Pa)



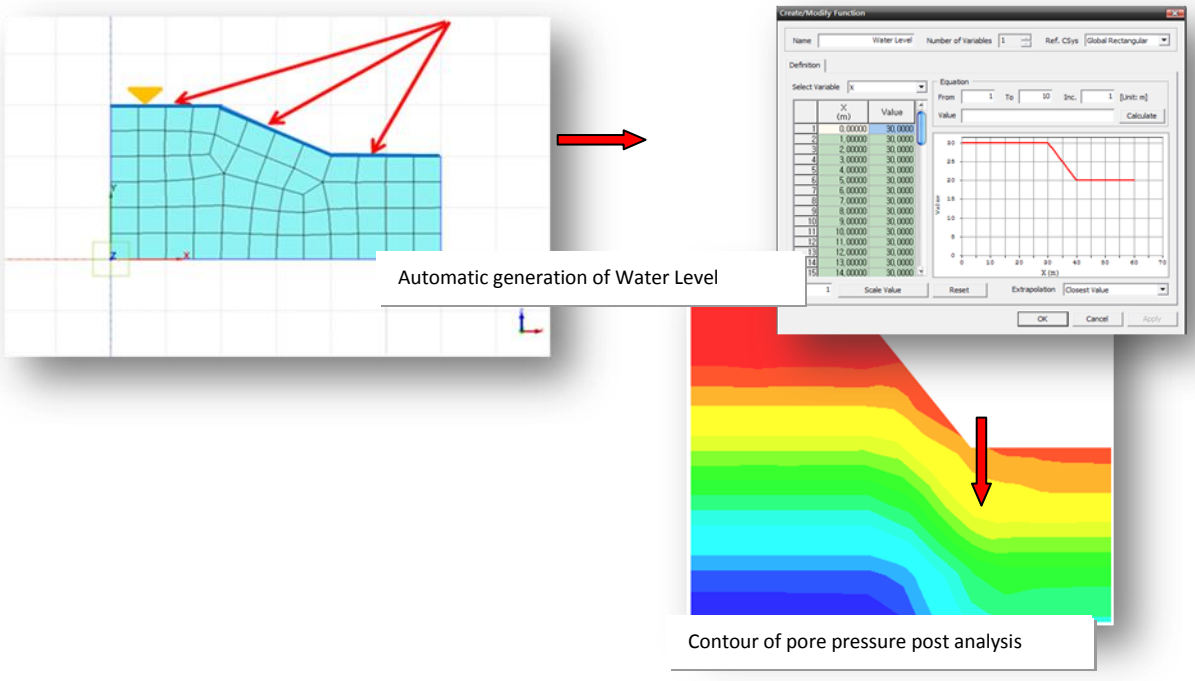
Tabular input for Material Properties

An additional feature has been added in GTS V300 that enables material property input through table format. The table is fully compatible with MS-Excel and users can edit the table for changing material properties.



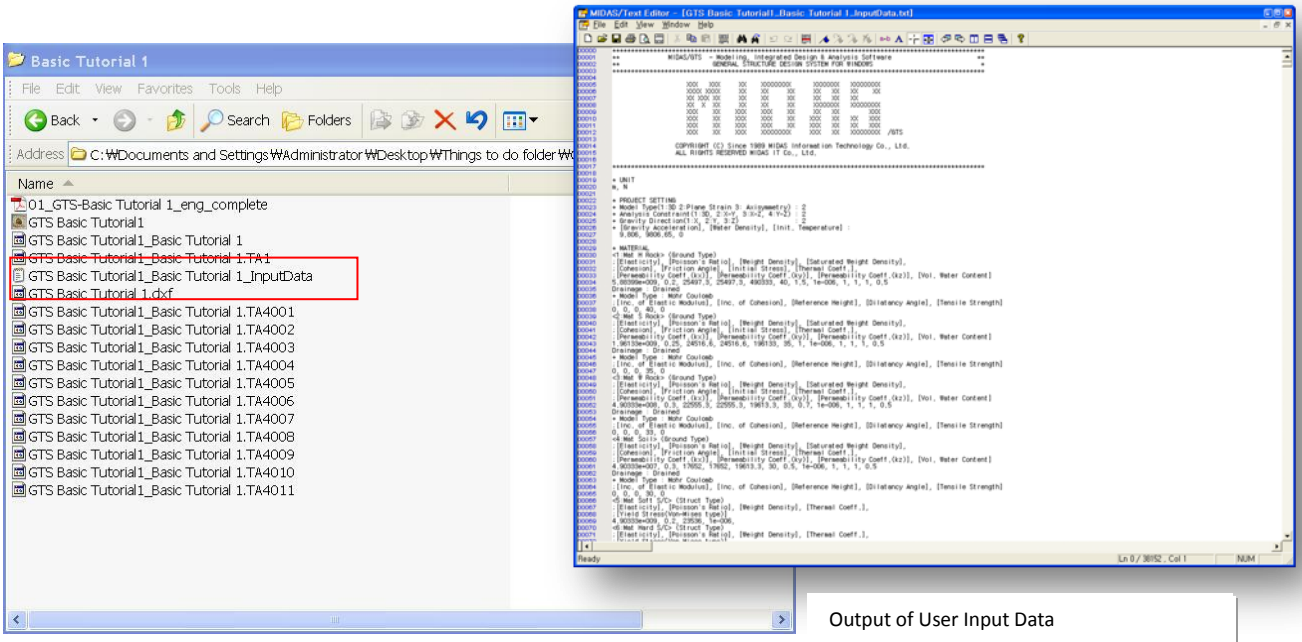
User Defined Phreatic Surface

A new feature that allows users to define phreatic surfaces in 2D by means of a geometry function. This can be done in the construction stage definition menu or separately through Model > Water level menu. The user can also select an existing geometry curve to be used as phreatic surface. In 3D, a constant water level can only be defined for the time being. However, it can be changed per construction stages.



Input Data ASCII File

User input data can be automatically saved as a *.txt format for reference. The file is generated at the end of the analysis and is referred to as *_InputData.txt.



Output *.txt file containing User Input Data

Output of User Input Data

Non-symmetric Multi-Frontal Direct Solver

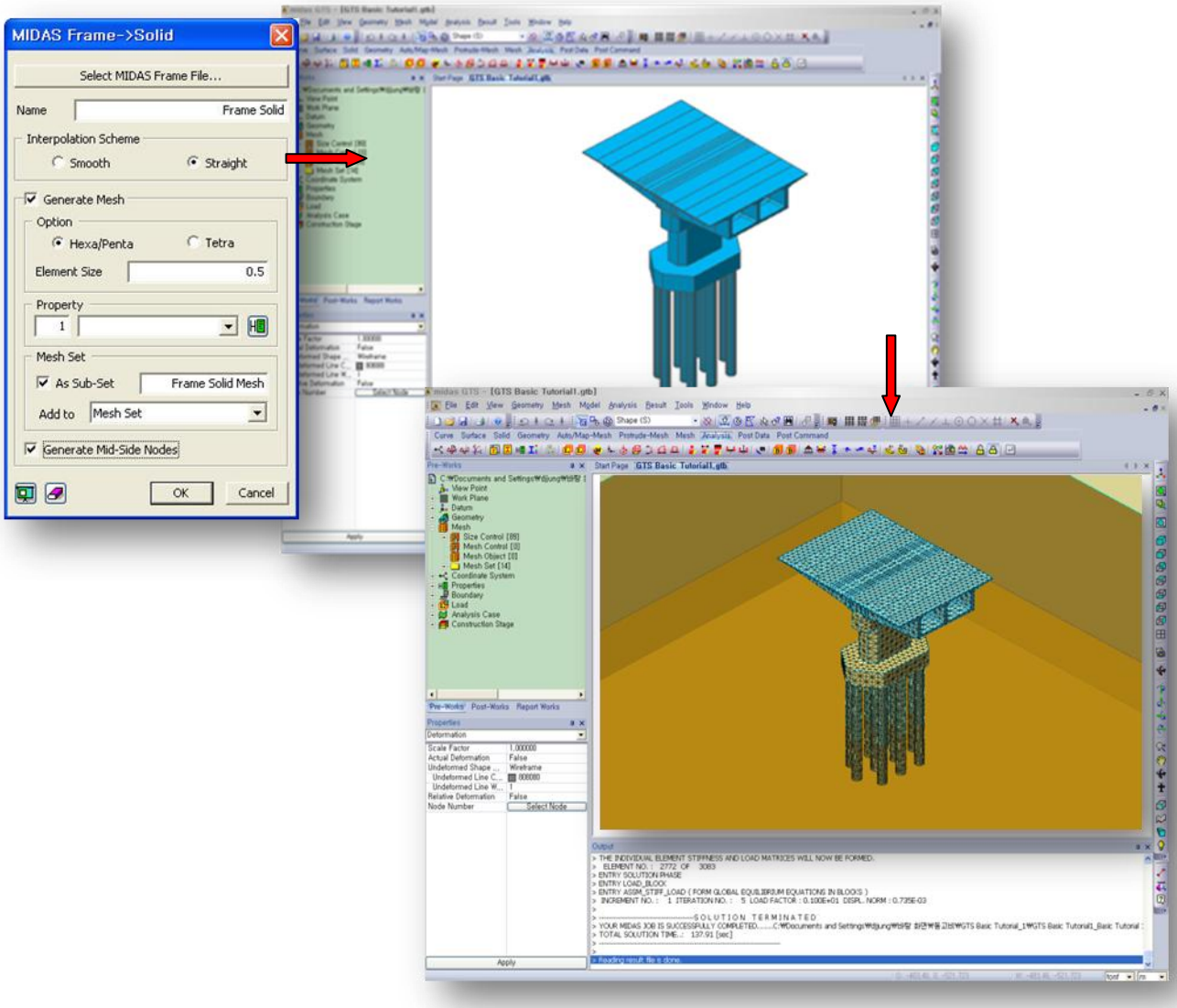
A standard Iterative solver is commonly used to handle linear systems with large sparse non-associated material models. However, it is cumbersome to predict whether the solver will converge or if it is capable of delivering accurate results due to the solver's reliance on tolerance settings. Moreover, iterative solvers are generally slow at converging or only converge for some cases. In order to provide better functionality, midas GTS has an enhanced Multi-Frontal Direct solver capable of handling not only associated but non-associated material models. The advantage in using a Multi-Frontal solver for non-associated material models can save users time while providing accurate results.

When the parameters, internal frictional angle and dilatancy angle are not equal, the Multi-Frontal Direct solver automatically performs analysis for a non-symmetric system. In short, the added enhancement in midas GTS provides a Multi-Frontal Direct solver and iterative operation for associated and non-associated material models.

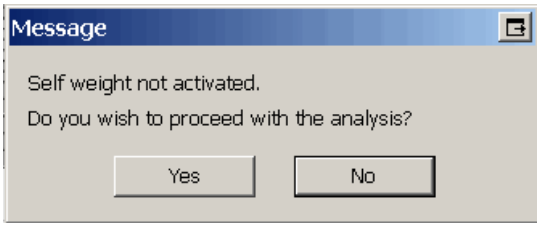
Improvements

File Exchange with midas Civil and midas Gen

Saved Frame data files from midas Civil and midas Gen can be converted in GTS from a frame to solid (Geometry> Frame -> Solid). This function converts the data into a geometrical or mesh interpretation. The frame to solid function contains an interpolation scheme providing users with smooth and straight functions. In addition, a Generate Mesh function contains three options for solid to mesh generation: Hexa/Penta, Tetra, and Element size.



Warning message regarding self-weight activation



In GTS V251, all analysis types can be analyzed regardless of the self-weight being activated or not. If the user forgot to activate self-weight, it was cumbersome for the user to re-run the analysis. This has been improved in GTS V300 so that a warning message notifies users about the status of the self-weight load after executing the Analysis > Solve command.

Linear Static Analysis for initial Construction Stage

In GTS V300, initial construction stage has been modified to only perform a linear static analysis to improve convergence.

Opening corrupted GTS files

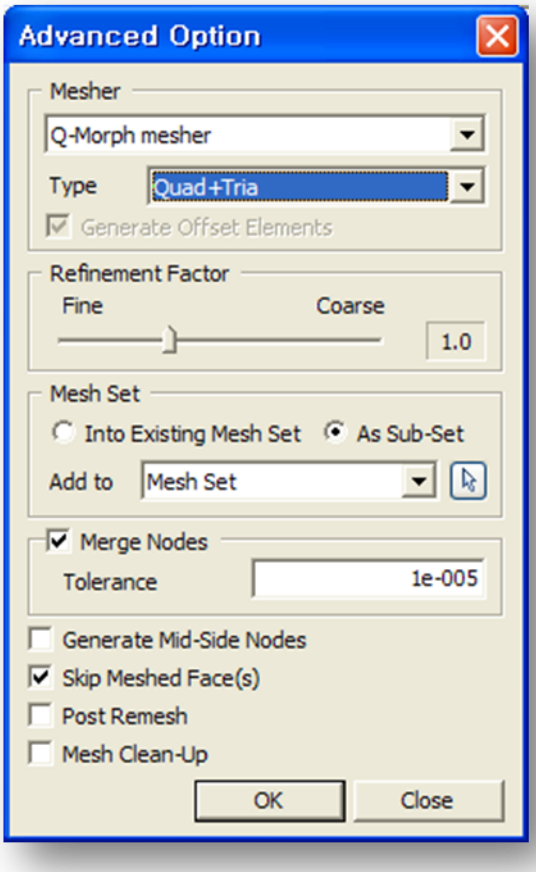
In GTS V300, saved model files containing entangled node connectivity can now be opened without importing the corrupted elements; and output the Element IDs and nodal information in the Output Window.

Truss/Embedded Truss

GTS V300 provides the following truss types when defining prestress:

1. Tension Only/Hook
2. Compression Only/Gap

Q-Morph Mesher



Quad-morphing is a technique that maintains all of the desirable features from paving algorithm and addresses some of its weakness. Quad-morphing uses an indirect method for generating quadrilaterals by taking advantage of local topology information from an existing triangle mesh. Beginning with an initial triangulation, triangles are systematically transformed and combined. An advancing front method is used to determine the order of transformations. An all-quadrilateral mesh containing elements, aligned with the area boundaries with fewer irregular internal nodes, can be generated. [1]

Advantages

1) Boundary Sensitive: Able to generate boundary sensitive rows of elements with few irregular nodes. Mesh contours closely follow the contours of the boundary Mesh.
2) Orientation Insensitive: The resulting mesh topology is unaffected by geometry rotation or translation.
3) Few Irregular nodes: Able to mesh with few irregular nodes near the boundary where the element shape

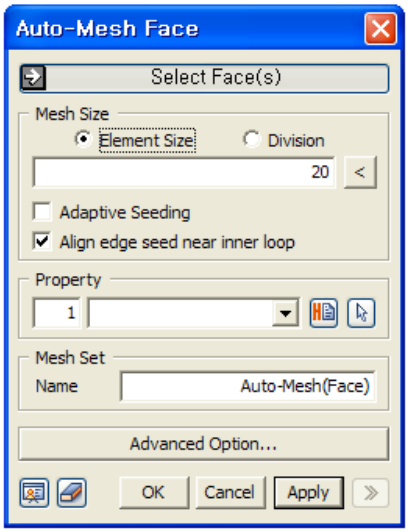
Disadvantage

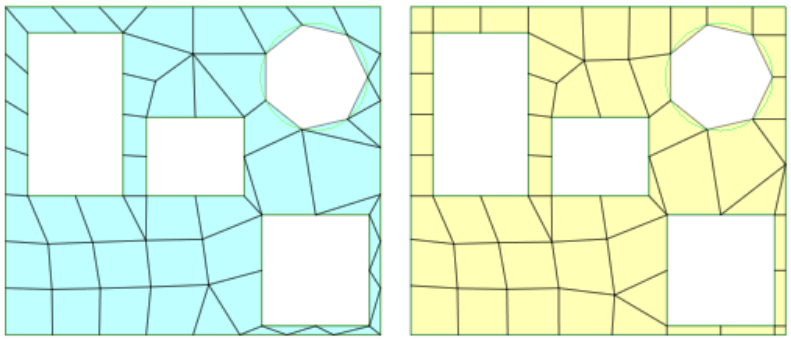
1) After generating the initial triangle mesh, the quadrilateral transformation processing time is longer, due to 3D surface mesh.

.

Align edge seed near inner loop

Improves the mesh quality, if an inner loop (hole) exists near the outer loop (boundary) of the face, edge seeding of the inner loop will align the edge seed near the inner loop with the outer loop edge.

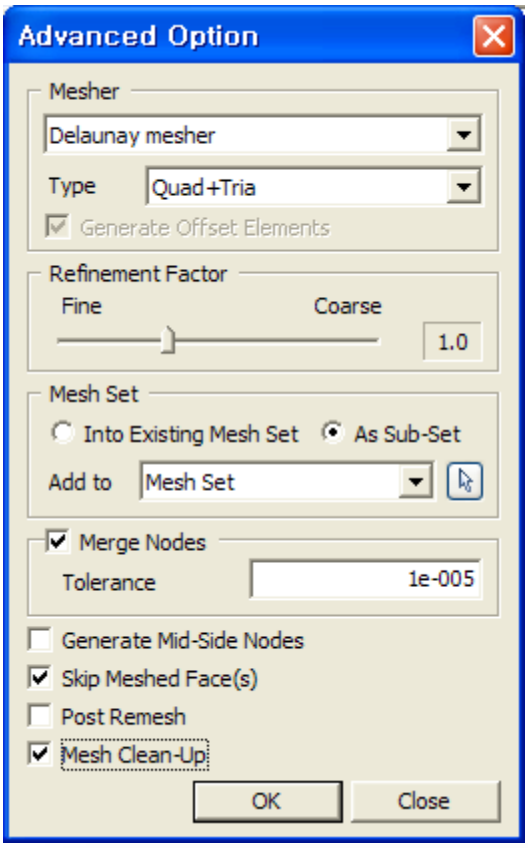




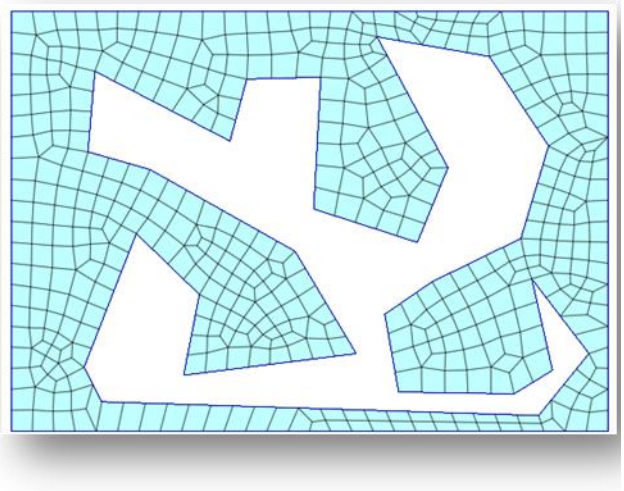
(a) (b)

Align edge seed near inner loop (a) Unchecked (b) Checked.

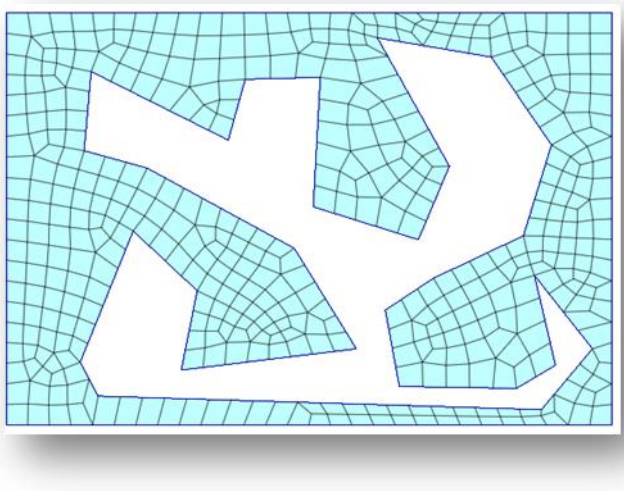
Mesh Clean-Up



Mesh->Auto Mesh Face ->Advanced Option->Mesh Clean-Up: Improves the quality of quadrilateral finite element meshes. Cleans all unused data from all selected mesh objects by improving node connectivity, boundary and flange patterns, quad shape, and to some extent, quad size.

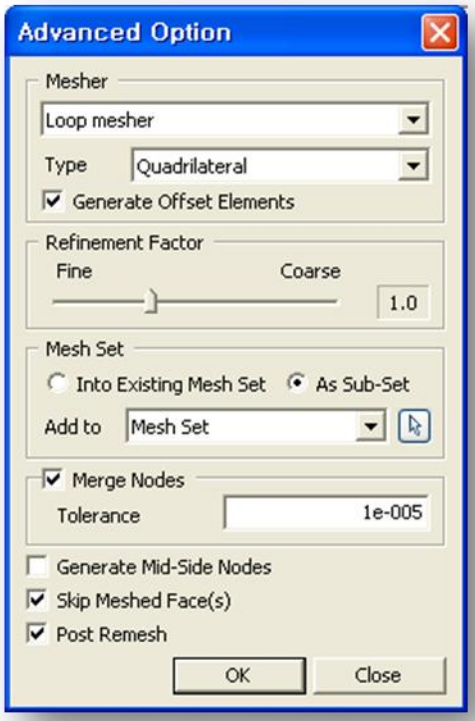


Option is checked off



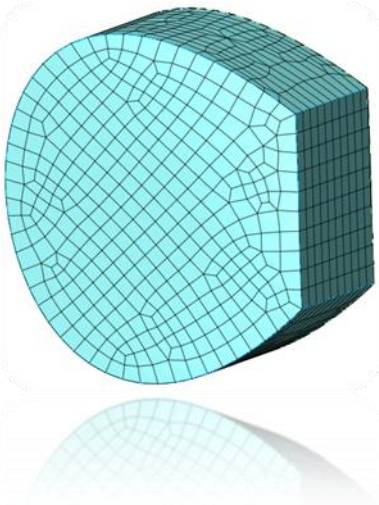
Option is checked on

Post Remesh

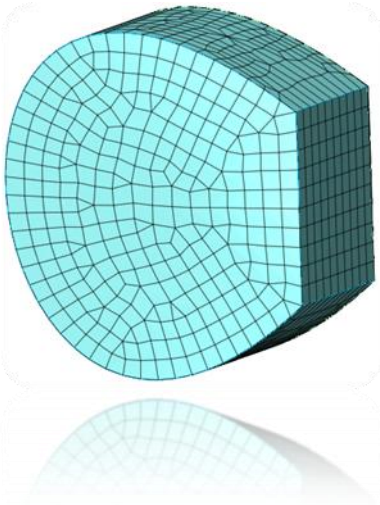


Menu : Mesh > 2D Mesh > Auto Mesh Face, Mesh > 3D Mesh> Auto Mesh Solid : Advanced Option

Post Remesh can be set before automeshing. When selected, it improves poor quality mesh from auto meshing. The existing mesh is deleted when the surface is remeshed

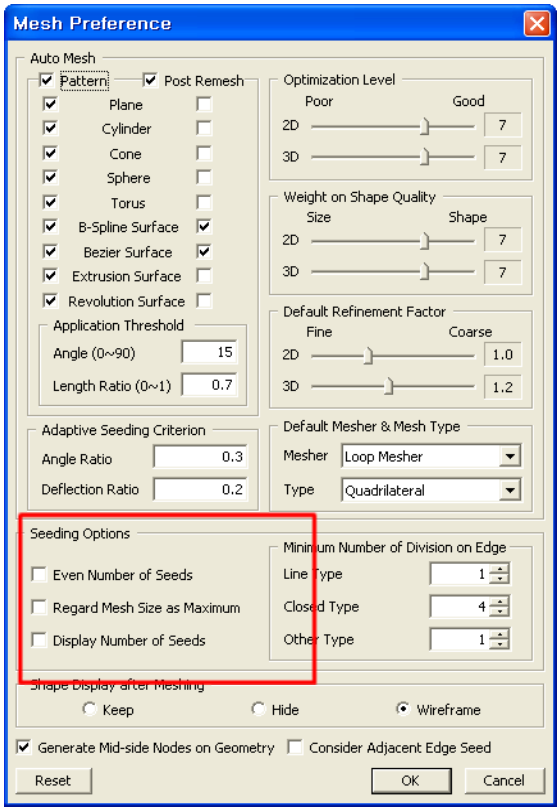


< without post remesh>



< with post remesh>

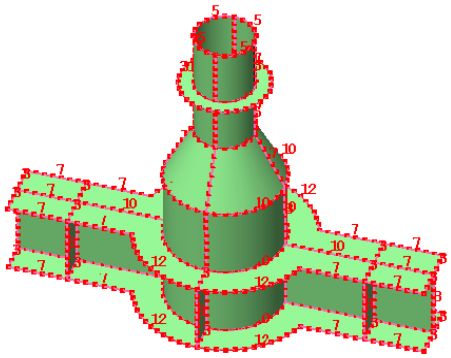
Improvements



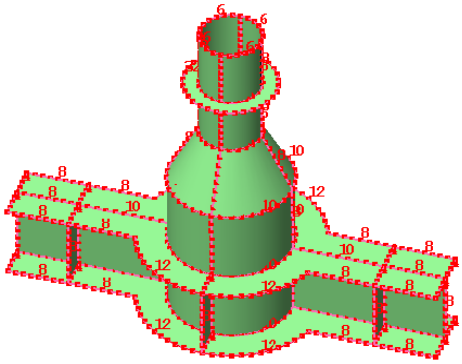
Mesh > Mesh Preference: Seeding Options

Even Number of Seeds:	Creates an extra node if elements contain odd number of nodes. Helps users to create quadrilateral elements.
Regard Mesh Size as Maximum:	Forces the number of nodes on an element to create an even number of nodes.
Display Number of Seeds:	Displays total number of seeds per edge.

Default seeds showing numbers



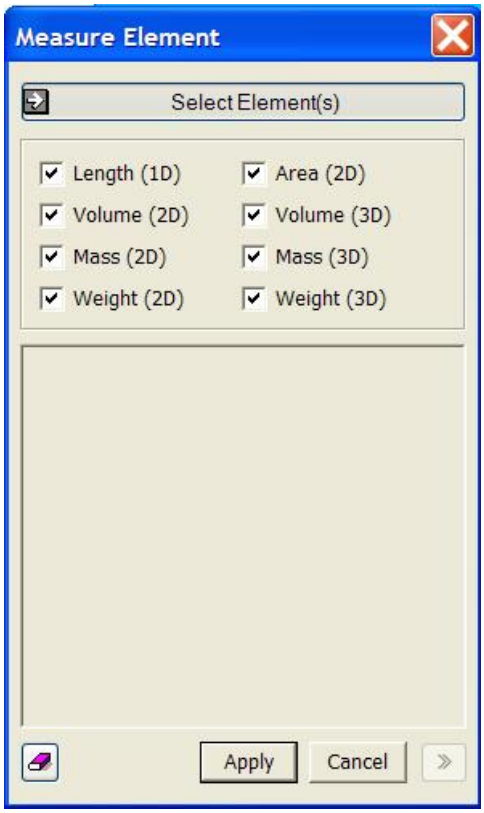
Seeds adopting even number option



Model > Element > Measure

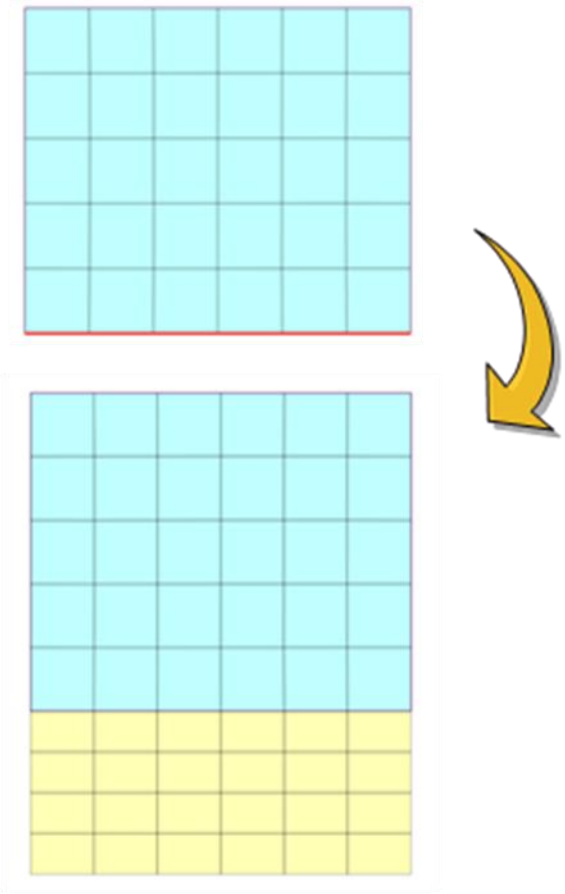
Description: This function allows users to measure element properties.

Dimension	Property
1D:	Length
2D:	Volume, Area, Mass, Weight
3D:	Volume, Mass, Weight



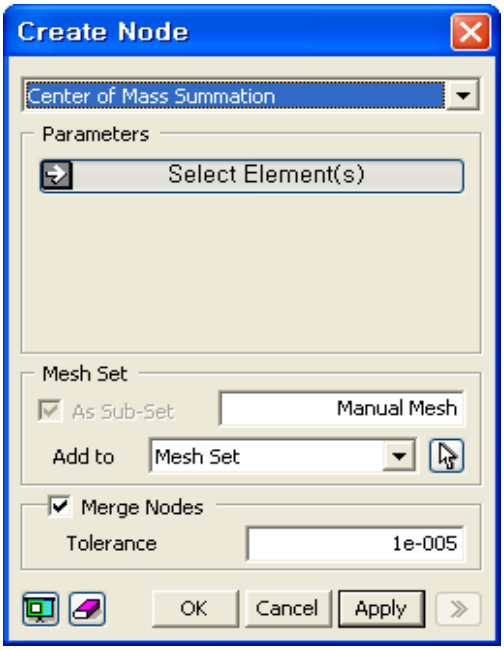
Mesh > Protrude Mesh > Extrude

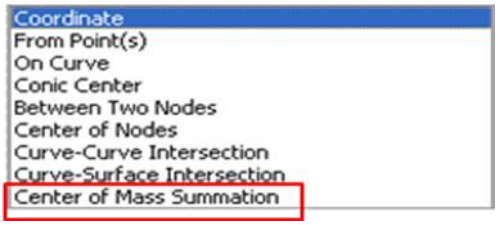
Description: Allows users to select an edge and extrude in the selected direction while controling the mesh length, offset/times, divisions.

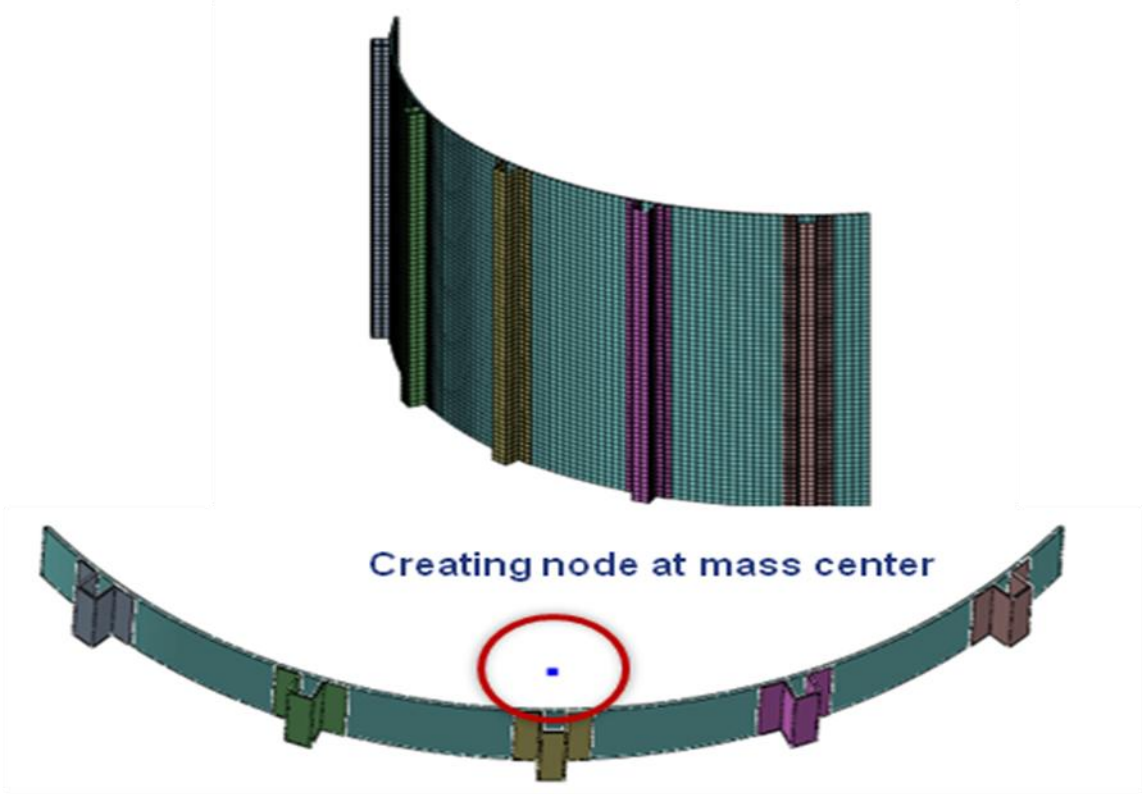


Model > Node > Create

Description: Creates a node using the center of mass summation function.

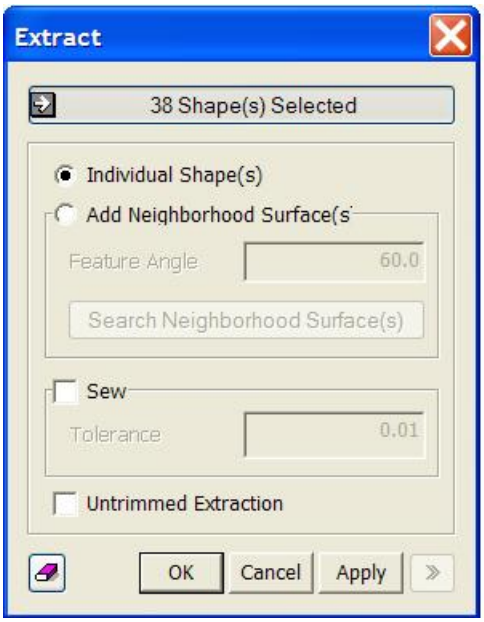


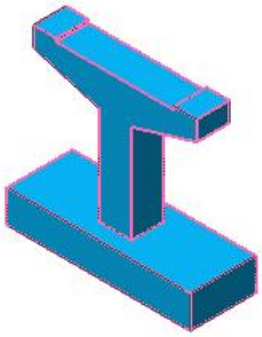


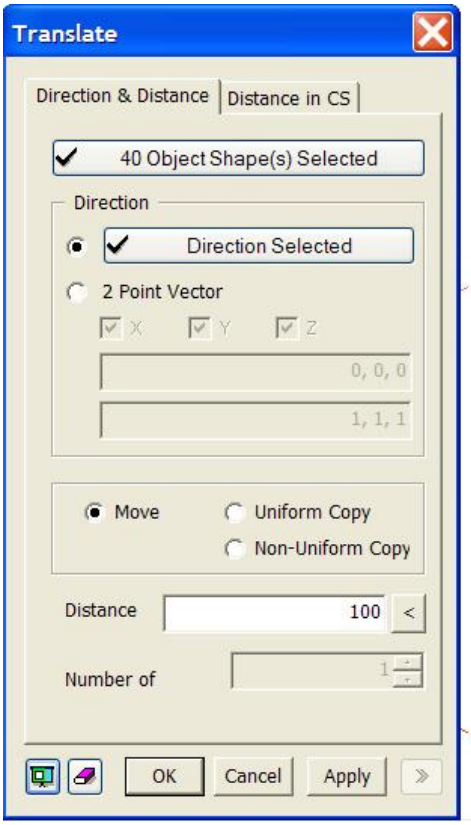


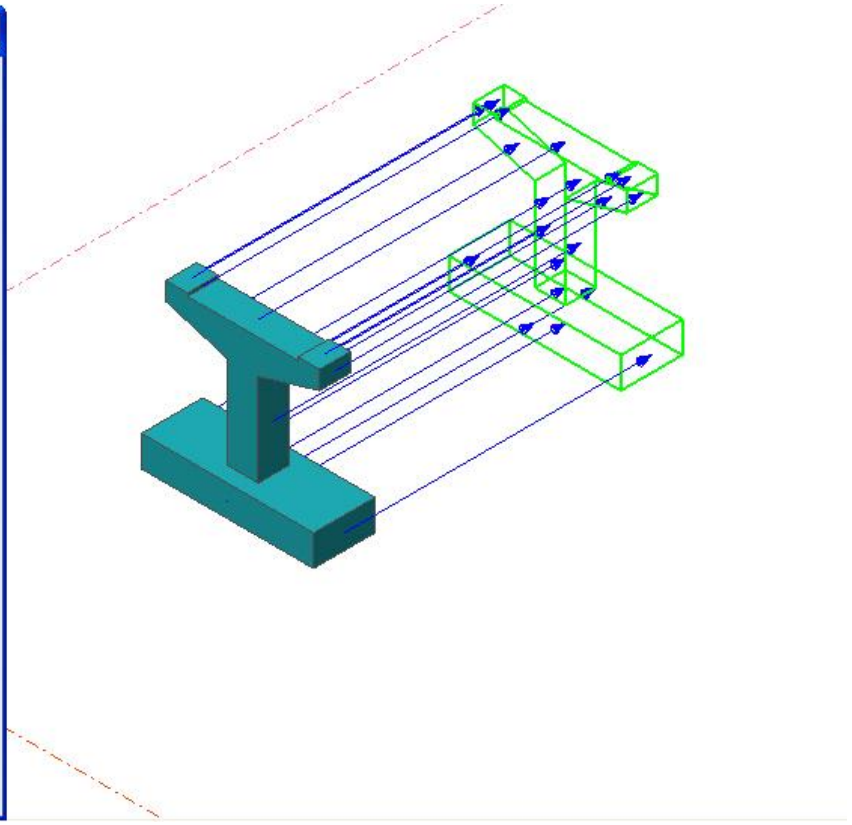
Geometry > Extract

Description: Allows the users to create surface only from a solid or 2D models.









GTS V300 Release Notes

Bug Fix: Analysis

1. Slope stability analysis with higher-order triangular shaped elements

Analysis> Analysis Case > **Slope Stability (SAM)**…

In GTS V251, an error occurred when users performed slope stability analysis using SAM method on higher-order triangular shaped elements. This has been resolved in GTS V300.

2. Deactivating prescribed displacement load in construction stage analysis

Model > Load > **Prescribed Displacement**…

In GTS V251, the prescribed displacement remained active despite having them deactivated in the construction stage definition. This has been resolved in GTS V300.

3. Incremental variation of Modulus of Elasticity and Cohesion with Depth

Model > Property > **Material**…

In GTS V251, when the reference height is defined anywhere in the model except ground surface for model types Linear, Drucker-Prager, and Mohr Coulomb, the material models' incremental parameters, such as modulus of elasticity and cohesion incorrectly applied above the reference line. This has been resolved in GTS V300.

4. Seepage Analysis

Model > Construction Stage > **Define Construction Stage**…

In GTS V251, when performing a combined Construction Stage and Seepage (steady state/transient) analysis, the seepage parameters, modified by Change Element Attributes, were not reflected in the analysis results. This has been resolved in GTS V300.

5. Stress-Seepage Semi-Coupled Analysis

Analysis > Analysis Case > **Construction stage**…

In GTS V251, for vertical infiltration and horizontal infiltration scenarios, the results from stress-seepage semi-coupled analysis were incorrect. This analysis calculated the effective stresses using pore-pressure by considering the hydraulic gradient. This has been resolved in GTS V300 by considering the variation of pore pressure from the Stress-Seepage Couple Analysis.

6. Stress initialization in construction-staged analysis

Analysis > Analysis Case > **Construction Stage**…

In GTS V251, at initial construction stage the internal solver failed to reset the initial strains to zero. This has been resolved in GTS V300. Both the displacement and strain values are set to zero in initial construction stage.

7. Linear Static Analysis

Model > Elements > **Create Interfaces**…

In GTS V251, the solver terminated when using 16 node plane interface elements. .This has been resolved in GTS V300.

Bug Fix: Analysis

8. Drucker-Prager Model

Model > Property > **Material**…

Analysis> **General Analysis Control**…

In GTS V251, when performing nonlinear analysis with the constant stiffness method, the solver did not recognize Drucker-Prager model's input parameters such as increment of modulus of elasticity and cohesion. This has been resolved in GTS V300.

9. Abnormal termination of Dynamic Analysis

Analysis > Analysis Case > **Time History (Linear)**

GTS V300 displays an error message appeared when performing a 2D or 3D dynamic analysis with pile type elements and/or interfaces.

10. Slope stability analysis

Analysis > Analysis Case > **Slope Stability**…

In GTS V251, when the slope is critically unstable that it can fail by self-weight, the program terminated. This has been resolved in GTS V300.

11. Transversely Isotropic & Jointed Rock Models

Model > Property > **Material**…

In GTS V251, when the option "Shear modulus" is not selected for Transversely Isotropic and Jointed Rock models, the shear modulus was incorrectly calculated. This has been resolved in GTS V300.

12. Strain results in Plate Output Coordinate System

Property >Property : Output Coordinate System

Post-Works > **Plate Strains**

In GTS V251, strain results for plate elements did not transfer properly to the Plate Output Coordinate System. This error did not occur for the force/stress results. This has been resolved in GTS V300.

13. 2D Geogrid Element Application

Model > Property > Property > **Plane Type: Geogrid (2D)**…

In GTS V251, the solver incorrectly calculated the stresses of 2D Geogrid elements. This has been resolved in GTS V300.

14. Lower/higher-order interface elements

Model > Elements > **Create Interface**…

In GTS V251, if one file contains two models with different order of interface elements, the program terminated when selecting results of the interface. This has been resolved in GTS V300.

15. Consolidation analysis Analysis > Analysis Case > **Consolidation**…

In GTS V251, the higher order plate and plane-strain elements modeled in the consolidation analysis type did not activate into the solver-causing it to terminate. This has been resolved in GTS V300.

16. 3D Geogrid element Analysis > Analysis Case > **Construction stage**…

An error occurred when 3D Geogrid elements were activated in the Construction Stage. The solver did not correctly calculate the stresses in between the linear and non-linear transition. This has been resolved in GTS V300.

17. Mohr-Coulomb Ko input parameter Model > Property > **Attribute**…

When enabling the Tensile strength for a Mohr Coulomb model type, the value 1.0 was assigned to any user-defined Ko input parameter. This has been resolved in GTS V300.

18. Seepage stress analysis Analysis > Analysis Case > **Construction stage**…

Computation of Seepage force did not consider water density. This has been resolved in GTS V300.

19. Surface Spring Model > Property > **Property**…

GTS V300 automatically constrains the rotational degrees of freedom. This has been resolved so that constraints can be only defined by the user.

20. Nonlinear analysis using high order plate and solid elements Analysis > **Solve**…

High order tetra elements used with higher order plate elements caused the solver to occasionally crash in nonlinear analysis. This has been resolved in GTS V300R3.

GTS V300 Release Notes

Bug Fix: Pre/post processing

1. Updating Units of Pile Element Model > Property > **Property**…

In GTS V251, *Pile elements'* units pertaining to Reference Depth and Slope of Friction-Rel Disp. curve did not update automatically. This has been resolved in GTS V300 so that the units for Reference depth and Slope of Friction-Rel. Disp. curve updates automatically.

2. Matrix Spring Model > Property > **Property**…

In GTS V251, when the Matrix Spring is considered in the Construction Stage, all spring coefficients returned the value zero. This has been resolved in GTS V300.

3. Display of Label Arrows Model > Load > **Pressure Load**…

In GTS V251, Label Arrows' direction for pressure loads displayed negative loading values incorrectly. The routine function has been resolved in GTS V300 so that the Label Arrows are displayed correctly.

4. Importing Pile Attributes Model > Property > **Attribute**…

In GTS V251 an error occurred when importing pile element attributes into a model. This has been resolved in GTS V300 so that users are able to import pile element attributes.

5. Display Material/Property of Interface Elements Model > **Display Material/Property**…

In GTS V251, differentiating interface elements with different material properties using Display Material/Property function did not work. This has been resolved in GTS V300 so that interface elements can be easily differentiated from a list of other properties.

6. Input of Water level pre/post Model > Construction Stage > **Define Construction Stage**…

In GTS V251, the units for water level value defined in 'Define Construction Stage' dialog box did not update automatically. This has been resolved in GTS V300 so that the units update and are reflected in the analysis.

7. Table format regarding input of Prestress Model > Load > **Prestress**…

In GTS V251, the prestress values could be modified from work tree > table; however, any changes made in the table did not update and reflect in the analysis. This has been resolved in GTS V300.

8. Change element property for 1D interface Model > Element > **Change Parameter**…

In GTS V251 changing the Lower-order 1D interface elements to higher-order was not possible and prompted an error message. This has been resolved in GTS V300.

Bug Fix: Pre/post processing

9. Modified Mesh Control Mesh > Size Control > **Display Mesh Seed**

Mesh > **Delete Control Data**

In GTS V251, errors resulted for the following three cases: editing of the edge seeds, displaying seeds for sections containing mesh seeding, and deleting most of the assigned mesh seeds using "Delete Control Data…" This has been resolved in GTS V300 so that any modification in the pre-processing Mesh Control is allowed.

10. Generating Elastic link using Surface Spring Model > Element > **Create Surface Spring**…

GTS V251 does not generate surface spring (elastic link types) at the element edges for higher-order elements. This has been resolved in GTS V300.

11. Results of deactivated interface elements Analysis >Analysis Case >**Construction Stage**

In GTS V251, when deactivating interface elements during the construction stage, the deactivated interface elements remained active in the results. This has been resolved in GTS V 300.

12. Saving Parametric Study Data Analysis > **Parametric Study**...

In GTS V251, an error occurred when opening saved parametric study files. This has been resolved in GTS V300.

13. Extract Results Result > **Extract Result**…

In GTS V251, when calling the *Extract Result* function for 1D beam elements, the table results displays the nodal coordinates in the z-direction incorrectly. This has been resolved in GTS V300 so that the table's algorithm properly outputs the extracted positions.

14. Report Generation output for plate/embedded truss elements Result > Result Summary >**Report Generation…**

In GTS V251, the report generation function failed to include plate/embedded truss elements results. This has been resolved in GTS V300.

15. Result component combination Result > **Results Component Combinations**…

In GTS V251, when defining a model with linear elements, users can use the function "Result component combination" to obtain a contour as an output for post-processing. However, using higher-order elements outputted the results as zero. This has been resolved in GTS V300.

16. von Mises stress Post Data > Solid Stresses : **von Mises**

In GTS V251, von Mises stresses of solid elements subjected to identical uniform pressure distribution on two orthogonal surfaces were very close to zero but not exactly zero due to a rounding error. This has been resolved in GTS V300.

Bug Fix: Pre/post processing

17. Defining von Mises properties for Beam Elements Model > Property >**Attribute**

When defining the material properties for Beam elements, none of the von Mises parameters were correctly transferred to the solver. This has been resolved in GTS V300.

18. Plate element's rotational results Model > Property >**Attribute**

In the post works tree window, the rotational degrees of freedom were not considered. This affected all the results for bending components. This has been resolved in GTS V300R3.

19. Local Direction Force Sum Result > **Local Direction Force Sum**

The Local Direction Force sum function was unable to generate results for a solid or plate element types. This has been resolved in GTS V300.

20. Nodal Prescribed displacement results Result > **Displacement**

Using node local axis coordinates, the assignment of prescribed nodal displacement only displayed displacement at 50%. This has been resolved in GTS V300.

21. Elastic Link Force Results Result > Result Tables > **Elastic Link Force**

Incorrect information about the Elastic Link Forces was displayed for some elements. This has been resolved in GTS V300.

22. Principal Stress Nomenclature **Post-Works**

The nomenclature for maximum and minimum principal stresses has been changed. Maximum and minimum principal stresses are now defined as P3 (V) and P1 (V), respectively.

23. Quadrilateral interface elements **Post-Works**

The results of quadrilateral interface element (low order and high order) did not display any contour plots. This has been resolved in GTS V300.

Known Issues

- 'Model>Load>Pressure Load>Edge pressure' is not applicable in 3D analysis. Loads along edges of solids should be specified using the function 'Model>Load>Force>Curve'.

- Note that no strain results are available in linear static analysis. Please use non-linear analysis with linear elastic material if strain results are required.

- Note that three-noded beam or truss elements are not available. When using 2D or 3D quadratic elements, first create 2-noded beams with the same element size as the adjacent quadratic elements. Then restore full nodal compatibility using 'Model>Element>Divide 1D Element' using Number of Divisions=2 and the Merge Node option selected.

- The rigid link property dialogue (in Spring/Interface tab) allows specifying rigid connections in translations (DX, DY, DZ) and in rotation (RX, RY, RZ). When rigid links connect two mesh sets with translational degrees of freedom (DOF) only, e.g. connecting plain strain elements or solid elements, only translational DOF should be activated in the rigid link property dialogue. If rotational DOF are specified in the rigid link definition, it will result in the analysis aborting. Similarly, for 2D analysis, only DOF in the analysis plane should be activated in the rigid link dialogue.