# GETTING STARTED

midas **FEA**

Advanced Nonlinear and Detail Analysis Program

# CONTENTS

## PART I. Getting Started

### 1. Introduction

1.1 Introduction ---------- Ⅰ-1-1

### 2. Installation

2.1 System Requirements ---------- Ⅰ-2-1

2.2. Installation Sequence – midas FEA ---------- Ⅰ-2-2

2.3. Installation Sentinel/Pro Driver ---------- Ⅰ-2-5

2.4. Registration the Protection Key ---------- Ⅰ-2-7

### 3. Understanding midas FEA

3.1 Before Getting Started ---------- Ⅰ-3-1

3.1.1 Work Sequence of midas FEA ---------- Ⅰ-3-1

3.1.2 Modeling Process of midas FEA ---------- Ⅰ-3-7

3.2. Organization of Window and Menu System ---------- Ⅰ-3-24

3.2.1 Work Window ---------- Ⅰ-3-27

3.2.2 Works Tree Window ---------- Ⅰ-3-28

3.2.3 Property Window ---------- Ⅰ-3-31

3.2.4 Output Window ---------- Ⅰ-3-33

3.2.5 Main Menu ---------- Ⅰ-3-37

3.2.6 Toolbar and Icon Menu ---------- Ⅰ-3-38

3.2.7 Context Menu ---------- Ⅰ-3-40

3.2.2 Works Tree Window ---------- Ⅰ-3-28

3.3. Selection and Manipulation of Work Window ---------- Ⅰ-3-41

3.3.1 Selection ---------- Ⅰ-3-41

3.3.2 View Manipulation Functions ---------- Ⅰ-3-60

# CONTENTS

3.3.3 Model Representation ---------- Ⅰ-3-64

3.3.4 Data Input Command ---------- Ⅰ-3-69

3.3.5 How to Use the On-line Manual ---------- Ⅰ-3-71

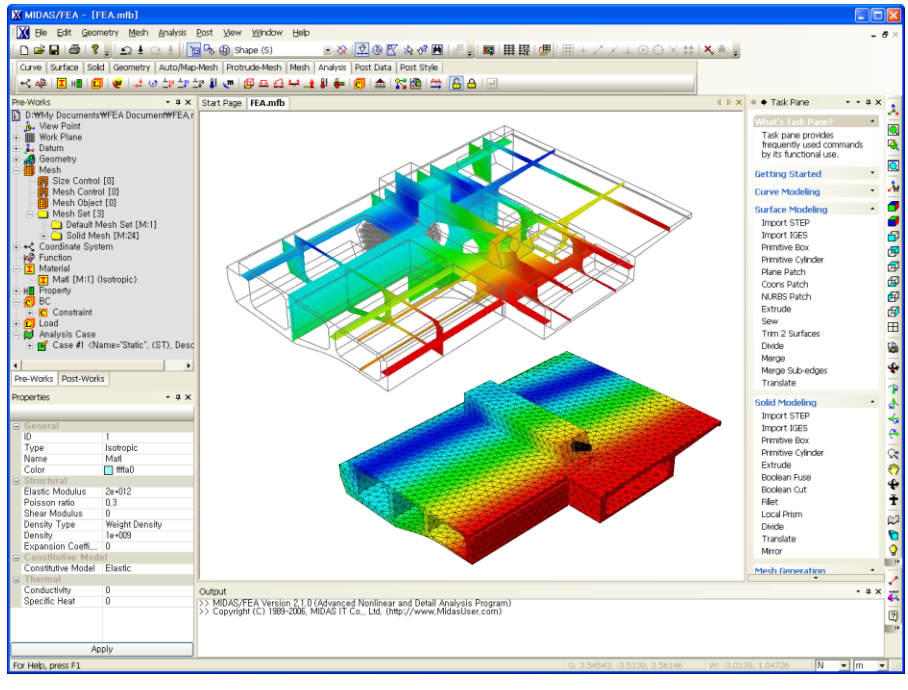
# Part Ⅰ Getting Started

# 1. Introduction

# 1.1 Introduction

midas FEA is state of the art software, which defines a new paradigm for advanced nonlinear and detail analysis of civil structures. With the experience in structural analysis software development since 1989, MIDAS IT delivers reliable solutions with utmost accuracy through the software.

In addition to exceptional analysis performance, midas FEA is equipped with advanced geometric modeling functions, powerful mesh generation algorithms, various analysis conditions and exceptional output displays with the latest graphics technology.




***midas FEA Environment***

# 2. Installation

# 2.1 System Requirement

midas FEA operates on IBM compatible Personal Computer (PC) in the Windows environment.

In addition, midas FEA requires the following minimum configuration.

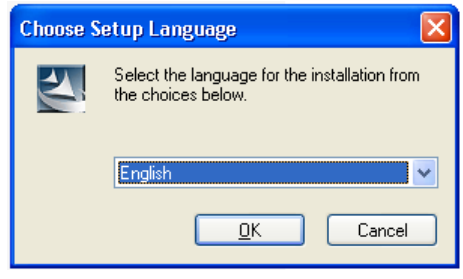
| | |
|---|---|
| ***Operating System*** | Microsoft Windows 2000 / ME / XP / Vista<br>NT 4.0 SP3 + Y2K Patches<br>(Microsoft Windows 2000 or later version recommended)<br>MS Internet Explorer 6.0 or higher |
| ***CPU*** | Pentium III 700MHz<br>(Pentium IV 1 GHz or greater recommended) |
| ***Memory (RAM)*** | 256 MB<br>(512 MB or greater recommended) |
| ***HDD Space*** | 1 GB |
| ***Video Memory*** | 32 MB or greater |
| ***Video Card*** | TNT/GeForce Type Videocard recommended<br>Windows XP / 2000 (30.82 Version or later)<br>➢ *http://www.nvidia.com/view.asp?IO=winxp-2k_30.82*<br>Windows NT ( 29.42 Version or later)<br>➢ *http://www.nvidia.com/view.asp?IO=winnt_29.42* |
| ***Printer*** | Windows compatible Printer or Plotter |

# 2.2 Installation Sequence - midas FEA

Follow the steps below to install midas FEA.

1. Insert the **midas FEA** CD in the CD-ROM drive.

2. If the ***Shift*** key is not pressed when the CD is inserted, installation will proceed automatically, and the **midas FEA** CD will process automatically. Select English for the setup language and click [OK] button.




***midas FEA Choose Setup Language Dialog Box***

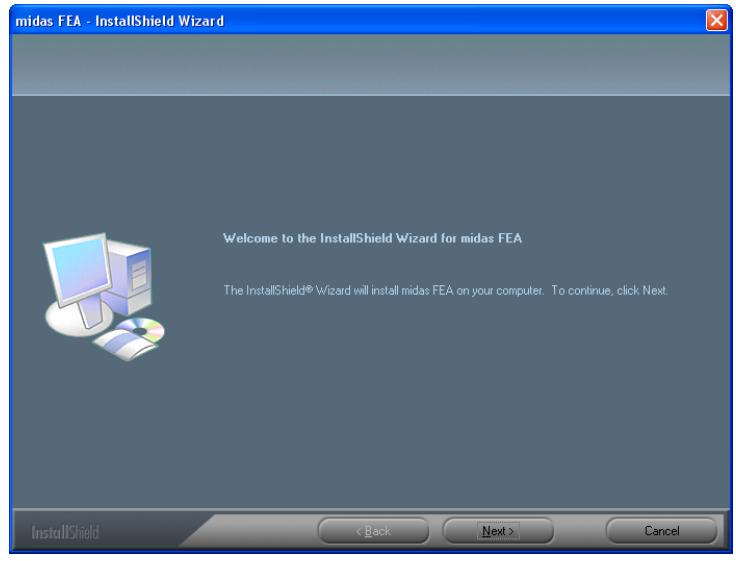
If the installation does not proceed automatically, select the ***Run*** command in the ***Start*** menu of Windows. Once the CD-ROM drive is assigned, enter the following command:

***D:\Install\Setup***

(Note: This applies in case the CD-ROM drive is assigned to the directory D)

Click [OK] to start the installation of the program.

3. Once the installation program is initiated, the dialog box shown below appears, and the installation of **midas FEA** begins. The installation will proceed step-by-step to the subsequent phases following the displayed information. To proceed to the next step, click Next >. To return to the previous step, click < Back.




***midas FEA Installation Dialog Box***

4. When the license agreement dialog box is displayed, read the agreement carefully. If you agree with the agreement, check on "I accept the terms of the license agreement". Click Next >, and the installation will continue.

5. Enter the user's registration information and click Next >.

6. The "Choose Destination Location" dialog box will appear. Select the folder in which **midas FEA** will be installed. **midas FEA** will be installed in the default folder by clicking Next >. To change the folder, click Change... and select a folder to install **midas FEA**.

7. The “Select Feature” dialog box will appear. To select all the features, click

and proceed. If not, uncheck unnecessary features, and click

.

8. To return to the previous step, click

. To start the installation, click

.

9. When the lock driver is installed, **midas FEA** installation will be completed.

# 2.3 Installation Sentinel/Pro Driver

The Sentinel Driver is used to drive the Lock key of the Sentinel hardware. To run midas FEA and the Lock key, the driver has to be installed. The Sentinel Driver is installed automatically during the installation process of midas FEA. For upgrading or replacing a damaged Lock driver, follow the procedure outlined below.

- To install the Sentinel Driver manually follow these steps.

1. Press the ***Shift*** key on the left side of the keyboard and insert the **MIDAS/Civil** CD in the CD-ROM drive.

2. Select the Run command in the Start menu. Once the CD-ROM drive is assigned, enter the following command:

   ***D:\Install\protection drivers\setup***

   (Note: This applies in case the CD-ROM drive is assigned to the directory D)

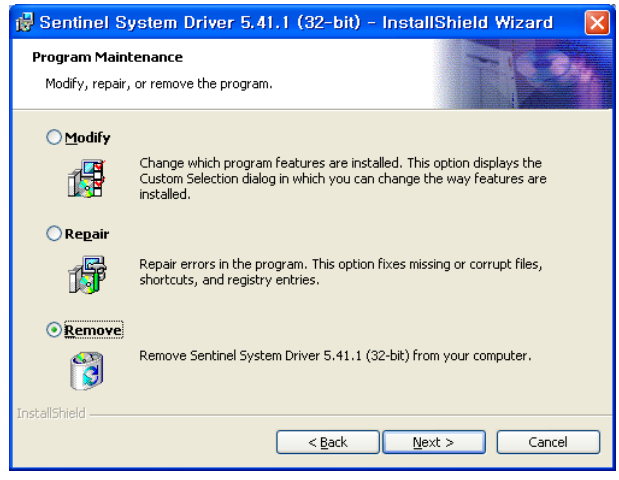
- To uninstall the Sentinel Driver follow these steps**.**

1. Press the ***Shift*** key on the left side of the keyboard and insert the **MIDAS/Civil** CD in the CD-ROM drive.
2. Select the ***Run*** command in the ***Start*** menu. Once the CD-ROM drive is assigned, enter the following command:

   ***D:\Install\Protection Drivers\SSD5410-32bit.exe***

   (Note: This applies in case the CD-ROM drive is assigned to the directory D)

3. In the Program Maintenance dialog box, select “Remove” to start uninstalling the lock driver.

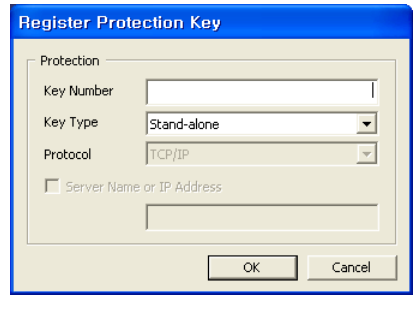



***Program Maintenance Dialog Box***

# 2.4 Registering the Protection Key

To operate midas FEA properly, register the serial number after connecting the protection key to the USB port.

1. Connect the Protection Key to the USB Port.
2. Execute **midas FEA**.
3. The ***Register Protection Key*** is located in the ***Help*** command of the ***Main Menu***.
4. Select ***Register Protection Key*** in the ***Help*** menu.
5. Enter the ***Protection Key ID*** provided in the Program CD Case in the ***Protection Key*** field.
6. Select 'Stand-alone' in ***Key Type***.
7. Click OK.




***Register Protection Key Dialog Box***

# 3. Understanding midas FEA

# 3.1 Before Getting Started

## 3.1.1 Work Sequence of midas FEA

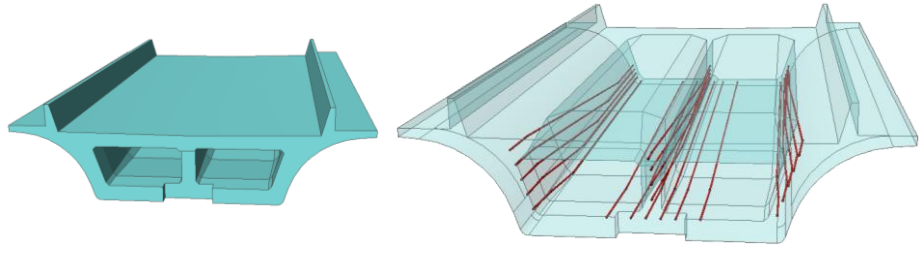
General work sequence of midas FEA is as follows:

**A. Geometry Modeling**
**B. Mesh Generation**
**C. Analysis Condition**
**D. Analysis**
**E. Post-processing and Result Evaluation**

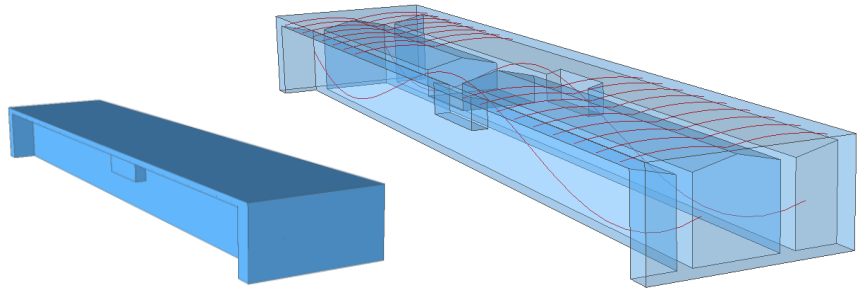
### A. Geometry Modeling

Generating a geometric model is the foundation for completing a finite element analysis model in midas FEA. Based on the geometry data, mesh generation and other additional modeling processes will ensue. The geometry model can be generated directly in midas FEA using its own modeling functions. In addition, midas FEA provides the data exchange capability, which enables import of a geometry data file created by other Computer Aided Design (CAD) software.

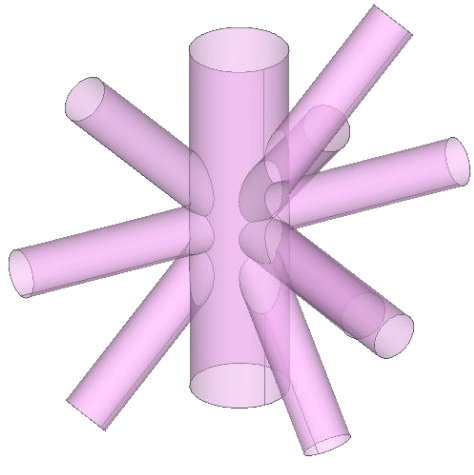
When a geometry data file is imported from another program, it often requires some special operations such as simplifying shapes, healing damaged geometry or cleaning-up disordered shapes in order to create a quality analysis model.



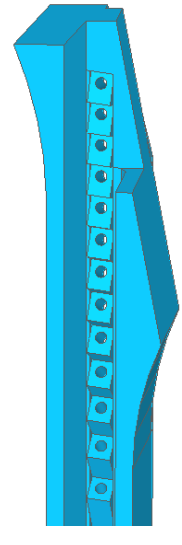
*Solid Model of PSC Bridge with Internal Tendons*



*Solid Model of PSC Girder with Internal Tendons*



*Shell Model of Pipe Joint*

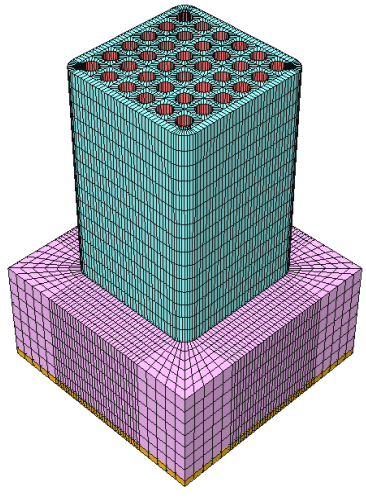


*Solid Model of Pylon*

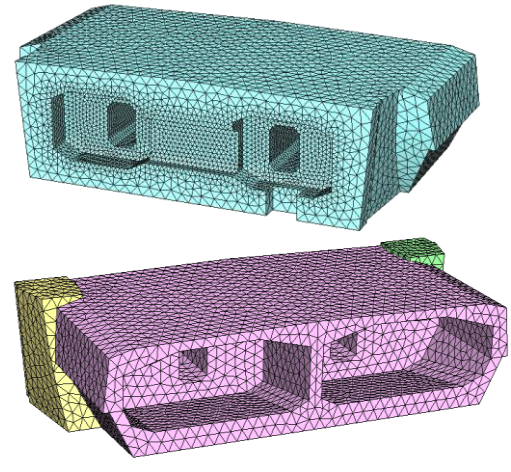
## B. Mesh Generation

Mesh is generated on a previously created geometry model. Generally, hexahedron elements or quadrilateral elements are recommended for meshing to attain accuracy in analysis results. However, for a complex model, it is reasonable to use tetrahedron elements or triangular elements produced by the Auto-mesh generation function provided by midas FEA.

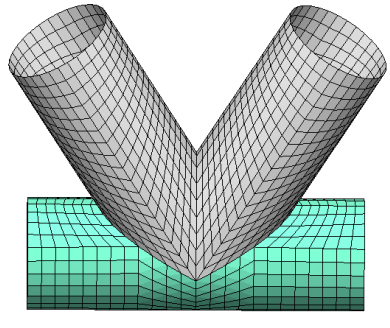
For best possible mesh quality, midas FEA provides various mesh control functions and 3 different mesh generation methods; Auto-mesh, Mapped-mesh and Protrude-mesh.



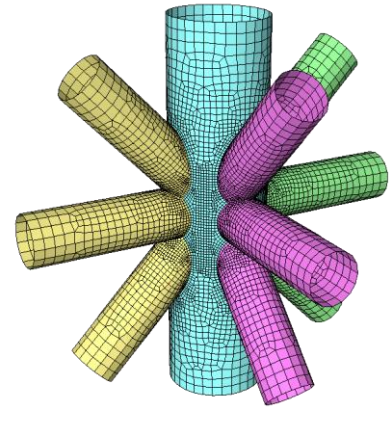
*Mapped-mesh Generated with Solid Hexa Mesh*



*Auto-mesh Generated with Solid Tetra Mesh*



*Mapped-mesh Generated with Surface Quad Mesh*

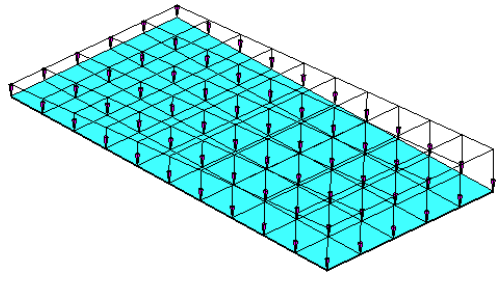


*Auto-mesh Generated with Surface Quad-dominant*

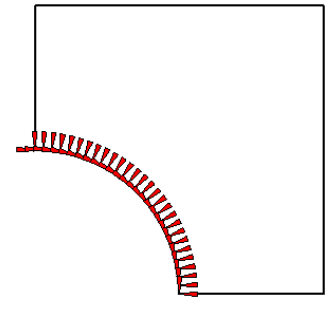
## C. Analysis Condition

midas FEA provides a variety of materials, physical properties, load types and boundary conditions. The load and boundary conditions can be applied not only to nodes and elements, but can be also directly applied to geometric shapes. Such capability is very effective when a model contains very complex shapes.

Taking advantages of the Windows based GUI environment, midas FEA maximizes the simulation capability to display the load and boundary conditions in a practical manner. As a result, the simulation function significantly reduces data input errors potentially committed by the user. All loads and boundary conditions can be created and modified from the GUI as well as from an MS-Excel compatible table.



*Example of Applying Non-uniform Pressure Load*



*Example of Applying Boundary Condition Using the Cylindrical Coordinate System*

## D. Analysis

Once the generated model is completed with proper analysis conditions, the analysis model can be analyzed. During the analysis, a variety of information such as progress and error messages is shown in the Output Window.

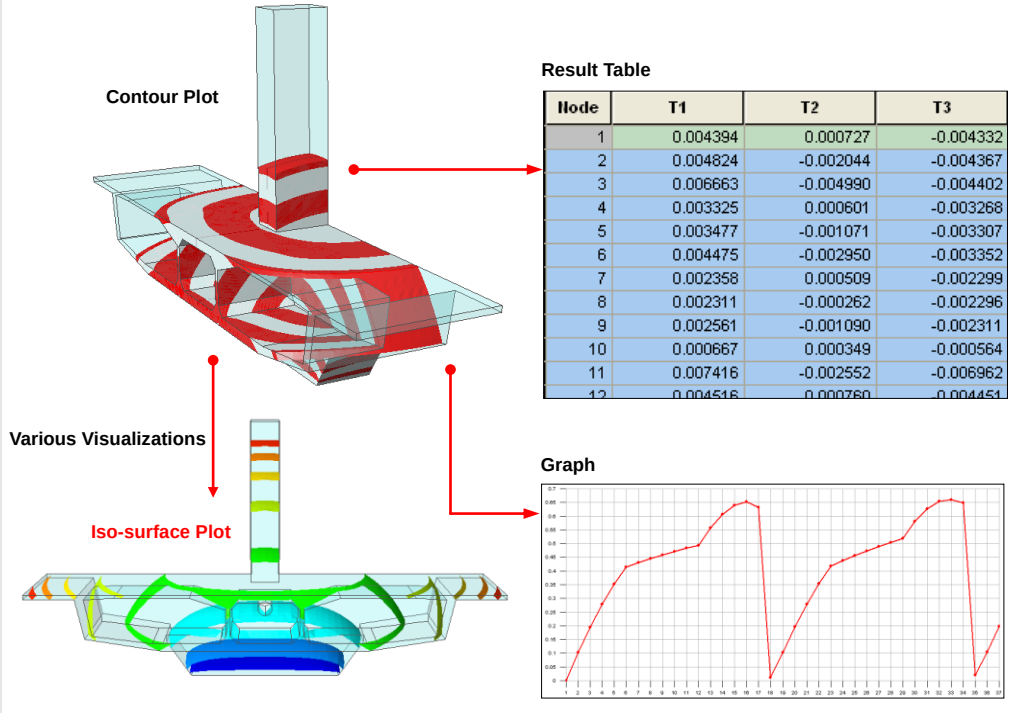
Based on the finite element method, midas FEA provides various analysis features such as Linear Static Analysis, Eigenvalue Analysis, Buckling Analysis, Time History Analysis, Response Spectrum Analysis, Material Nonlinear Analysis, Geometry Nonlinear Analysis, Concrete Cracking Analysis, Fatigue Analysis, Linear/Nonlinear Heat Transfer Analysis and Construction Stage Analysis.

In addition, midas FEA is equipped with a Multi-frontal Gaussian Sparse Solver and an Iterative Solver, which are known for prominent analysis speed in large models.

### E. Post-processing and Result Evaluation

Once the analysis is completed properly, midas FEA will organize and provide the post data results for design process. midas FEA offers numerous outstanding graphical result displays including the facility for creating high quality animations. In addition, all the tables provided in midas FEA allow the user to view the analysis results at a glance, and they are compatible with MS-Excel. The revolutionary post-processing capability of midas FEA will thus define a new standard for post-processors in the Computer Aided Engineering (CAE) industry.

In addition, using the Report Summary function, all modeling and result data will be organized into a practical report so that the user can maximize efficiency for his work.




| Node | T1 | T2 | T3 |
|---|---|---|---|
| 1 | 0.004394 | 0.000727 | -0.004332 |
| 2 | 0.004824 | -0.002044 | -0.004367 |
| 3 | 0.006663 | -0.004990 | -0.004402 |
| 4 | 0.003325 | 0.000601 | -0.003268 |
| 5 | 0.003477 | -0.001071 | -0.003307 |
| 6 | 0.004475 | -0.002950 | -0.003352 |
| 7 | 0.002358 | 0.000509 | -0.002299 |
| 8 | 0.002311 | -0.000262 | -0.002296 |
| 9 | 0.002561 | -0.001090 | -0.002311 |
| 10 | 0.000667 | 0.000349 | -0.000564 |
| 11 | 0.007416 | -0.002552 | -0.006962 |
| 12 | 0.004516 | 0.000760 | -0.004451 |

*Example of Various Post Processing features of midas FEA*

## 3.1.2 Modeling Process of midas FEA

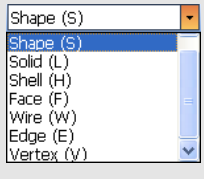
midas FEA allows the user to create highly sophisticated geometrical data and generate mesh with any dimension or shape. Especially due to its advanced geometry modeling tools and high-performance auto-meshing functions, the user can model a complex shape much quickly and simply compared to traditional modeling programs, which mostly pertain to manual node and element generation methods.

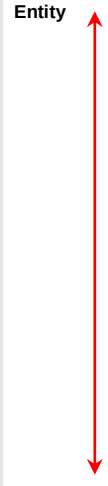
In this chapter, the geometry modeling and mesh generation will be discussed in detail.

### A. Geometry Modeling

A geometry model is fundamentally formed by the interlocking relationship of various geometric entities. The geometric entities in midas FEA are listed in the following table.

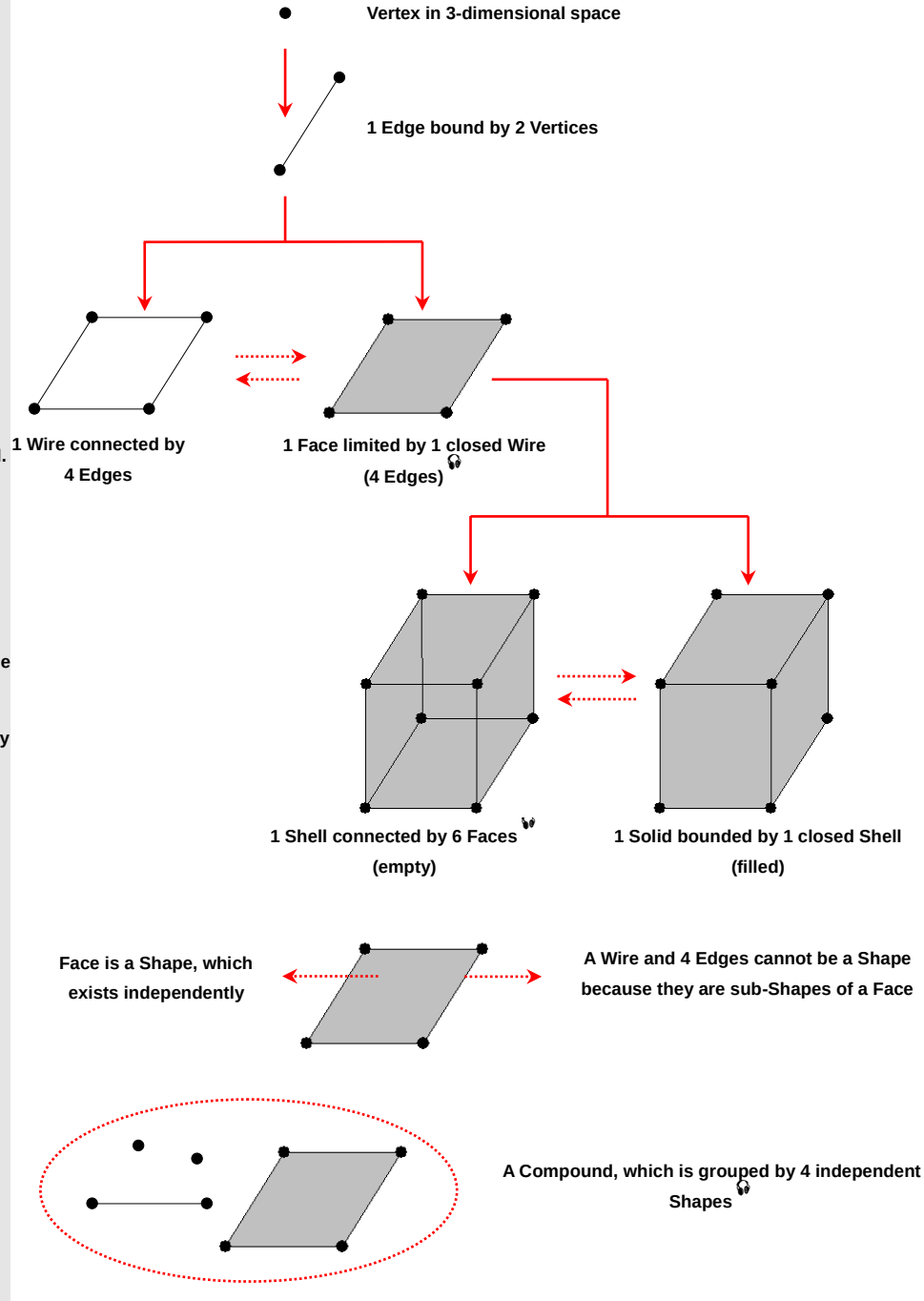
**Filtering enables us to select shapes by geometry types.**





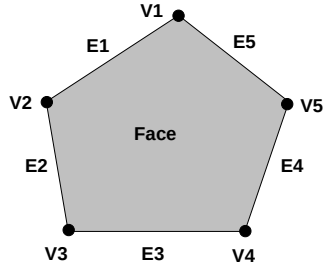

| Entity | | Definition | |
|---|---|---|---|
| Compound | | Group of independent entities | |
| Shape | | General term to call an independent entity<br>(An entity that is not forming any other entity) | |
| Solid | | Part of 3-D space bound by a shell | |
| | | Property | Volume |
| Surface | Shell | A collection of Faces connected by some of the Edges of their Wire boundaries | |
| | Face | Part of a plane (in 2-D geometry) or a surface (in 3-D geometry) bounded by a closed Wire<br>e.g. Plane surface, cylinder, sphere, etc. | |
| | | Property | Area |
| Curve | Wire | A sequence of Edges connected by their Vertices | |
| | Edge | A shape corresponding to a curve bound by a Vertex at each extremity<br>e.g. Straight line, circle, ellipse, arc, etc. | |
| | | Property | Length |
| Vertex | | A zero-dimensional shape corresponding to a point in geometry | |
| | | Property | Coordinate |

- The user may interchange freely between closed Wire and Face or between closed Shell and Solid because they share the same sub-Shapes.
- The boundary of a Face consists of one Wire, and the boundary of a Solid consists of one Shell.
- Extruding an Edge creates a Face. Extruding a Wire creates a Shell. This Shell shares the same sub-Shapes with a group of Faces, which is generated by the extrusion of the original Wire's sub-Edges.
- Compound is generally used to sort shapes easily. (i.e. grouping hundreds of Curves imported from Autocad's .dxf file into a single Compound.)



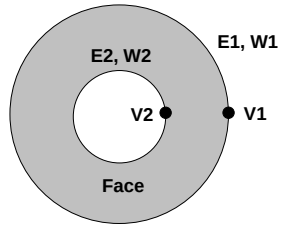
**Topology means the relationship of geometric entities.**

The following examples will help us understand the topology more comprehensively.



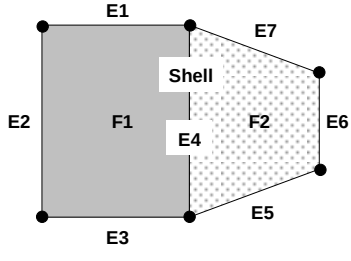

| Highest Shape | | One (Face) |
|---|---|---|
| Sub-Shape | Wire | One (Face's Boundary)<br>➢ Group of E1 through E5 |
| | Edge | Five (E1~E5) |
| | Vertex | Five (V1~V5) |

**No matter how many edges exist, a boundary of a Face will be always regarded as a Wire. In a closed Edge like a circle, starting and ending vertices are identical.**

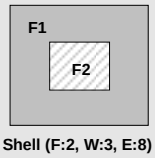



| Highest Shape | | One (Face) |
|---|---|---|
| Sub-Shape | Wire | Two (Inner/outer boundary)<br>➢ same as E1, E2 |
| | Edge | Two (E1, E2) |
| | Vertex | Two (V1, V2)<br>➢ Starting and ending points of edges |

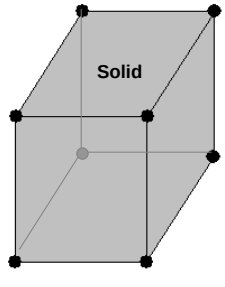
**When the user sews two neighboring Faces into a Shell, one or more Edge(s) will be shared between Faces. In the diagram, two independent Faces have a total of 8 edges, but the combined Shape, a Shell, has only 7 edges.**




| Highest Shape | | One (Shell) |
|---|---|---|
| Sub-Shape | Face | Two (F1, F2) |
| | Wire | Two (Boundary of each face) |
| | Edge | Seven (E1~ E7)<br>➢ They share E4 |
| | Vertex | Six |




Shell (F:2, W:3, E:8)




| Highest Shape | | One (Solid) |
|---|---|---|
| Sub-Shape | Shell | One (Solid's Boundary) |
| | Face | Six |
| | Wire | Six (Boundary of each face) |
| | Edge | Twelve<br>➢ 4 for each face, 12 edges are shared |
| | Vertex | Eight |

midas FEA provides both ***Top-down Modeling*** and ***Bottom-up Modeling*** methods. Therefore, the user can choose either method or a combination of both depending on the beneficial characteristics of a particular model.

- **Top-down Modeling Method**

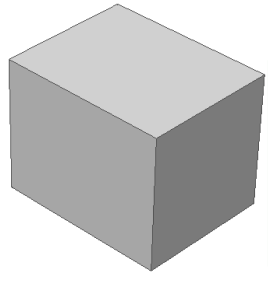
  In this method, the user makes the highest level Shape, and its sub-Shapes will be automatically formed. The user does not need to waste a lot of effort to model each sub-Shapes. The method is often used for a simply shaped model or for a preliminary design process.

  *i.e. If a cube is created with the primitive box feature of midas FEA, all sub-Shapes of this solid such as Faces, Wires and Edges are automatically generated.*

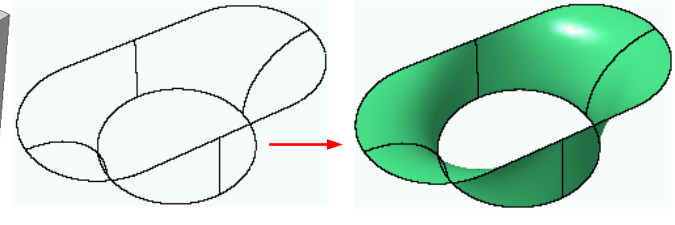
- **Bottom-up Modeling Method**

  In this method, the user starts to model from the lowest level entities first. With these independently generated sub-Shapes, the final shape is assembled together. The operation may take a lot longer time than the Top-down method. However, by this method, the user can generate a very complex shape, which cannot be readily created by the other method. In most geometry modeling, the Bottom-up Modeling Method is widely used in practice.

  *i.e. A 3D curved surface is extremely difficult to model directly at once. Therefore, Vertices and Edges, which represent the Face, will be created first, and the Face will be generated by using those Vertices and Edges.*



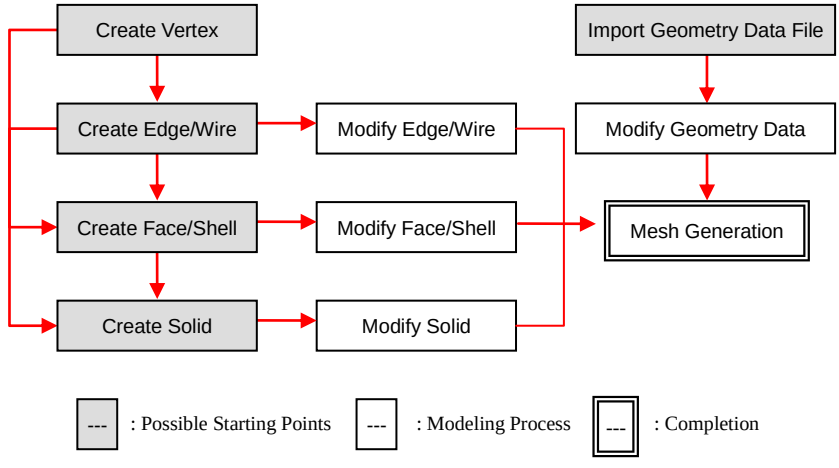
*A Box Creation by the Primitive Box Function*
*(Top-down Modeling: Solid)*



*3D Surface Generation with the NURBS Face*
*(Bottom-up Modeling: Edge→Face)*

It is not necessary to start modeling from the Vertex for the Bottom-up Modeling Method. If possible, the mid-steps between the bottom shape and the top shape can be eliminated.

The actual modeling process of the Bottom-up Modeling is shown below.




***Flow Chart of Geometry Modeling Process of midas FEA***

## B. Mesh Generation

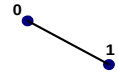
midas FEA provides various types of elements. Each element type is defined by the structural characteristic of its associated nodes.

**Element types are categorized by their geometrical characteristics, and the structural behaviors of elements may be different within the same type. (e.g. Truss and Beam) The structural behaviors can be found in the Analysis Reference Manual.**

- **Scalar Element**

  It consists of 1 or 2 node(s), and it does not have a geometrical characteristic. Mass, damper and spring elements fit into this category.

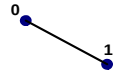
*1-Node Scalar*
*(e.g. Node Spring)*



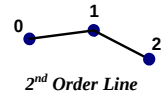

*2-Node Scalar*
*(e.g. Element Spring)*

- **1D Element (Line)**

  A 1-dimensional element has a geometrical characteristic of length and consists of 2 or 3 nodes. Truss and beam elements fall under this category.



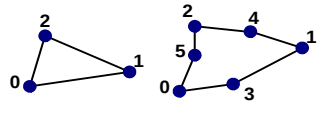

*1st Order Line*



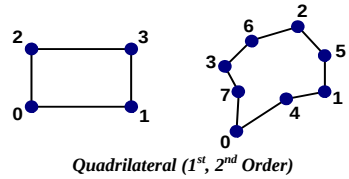

*2nd Order Line*

- **2D Element (Plane)**

  A 2-dimensional element, which represents either a triangular or a quadrilateral element, has the geometrical characteristic of area. Plane Stress Element, Plain Strain Element, Axisymmetric Element and Plate Element belong to the 2D Element type.



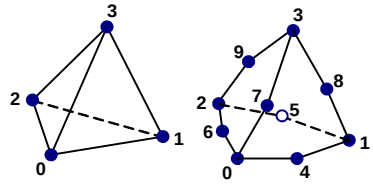

*Triangular (1st, 2nd Order)*



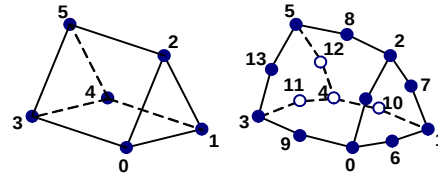

*Quadrilateral (1st, 2nd Order)*

- **3D Element (Solid)**

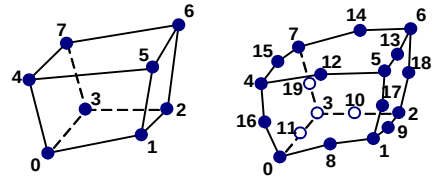
  A 3-dimensional element such as Tetrahedron, Pentahedron (Wedge or Prism) and Hexahedron (Brick) has the geometrical characteristic of volume.




*Tetrahedron ($1^{st}$ , $2^{nd}$ Order)*




*Pentahedron ($1^{st}$ , $2^{nd}$ Order)*




*Hexahedron ($1^{st}$ , $2^{nd}$ Order)*

- **Miscellaneous**

  A few element types such as Rigid Links are categorized here.

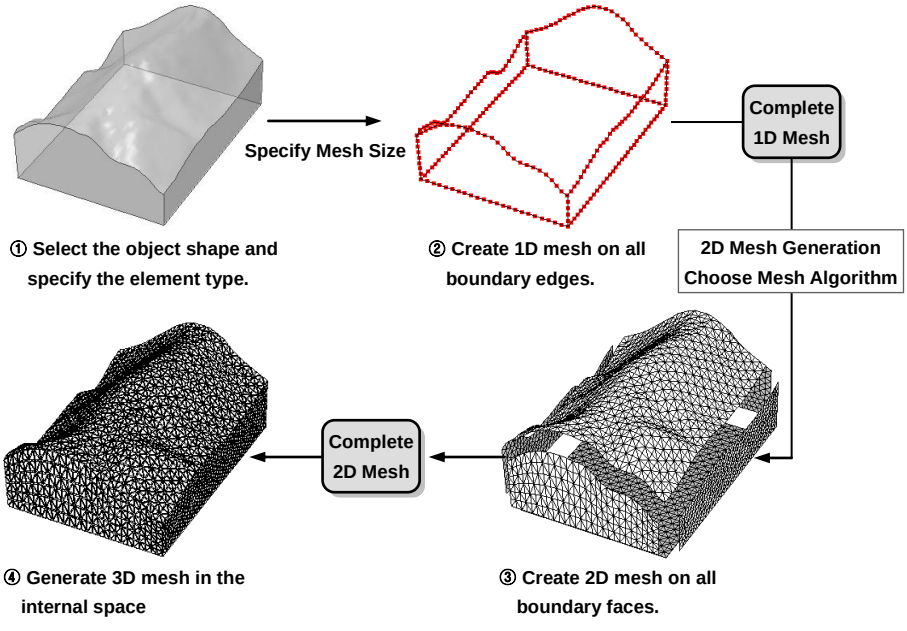
A structure in real life always exists in space, and it can be modeled by certain types of elements with proper dimensions. Its structural behavior can be expressed by an accurate material model with the corresponding geometric property. In order to construct an analytical model of a structure, its elements must include these 3 important characteristics; (a) nodal connectivity, (b) material property and (c) additional geometric property such as thickness.

The geometric properties, which need to be defined for each of 1D, 2D and 3D elements, are listed in the table below.

**The table to the right is intended for understanding the basics of the geometric property. The list only contains the common items. Depending on its structural behavior, each element type may require additional properties. For**

| Element Type | Geometric Characteristic | Property | Volume Calculation |
|---|---|---|---|
| 1D | Length(L) | Area | L × A |
| 2D | Area(A) | Thickness | A × t |
| 3D | Volume(V) | N/A | V |

Regardless of dimension and type, the general process of mesh generation is shown in the following diagram:




***General Process of Mesh Generation***

The detailed generation processes for each of 1D, 2D and 3D Mesh are as follows:

- **1D Mesh**
  One dimensional elements are generated on the selected edges based on the specified mesh size.

- **2D Mesh**
  Before generating 2D mesh, 1D mesh is produced on the boundary edges. The program executes this automatically in a hidden process. Subsequently, 2D mesh fills in the internal area, which is bounded by the previously generated 1D mesh. The 2D meshing algorithm can be specified either by the user or automatically by the program.

- **3D Mesh**

  After generating 1D and 2D mesh, the program generates 3D elements in the internal space, which is enclosed by the 2D boundary mesh. If necessary, 1D and 2D mesh generations are processed internally by the program.

  The user must use caution for the listed items prior to mesh generation.

  1. In order to generate a proper 2D mesh, 1D mesh on the boundary edges must be completely enclosed. Similarly, in order to generate a proper 3D mesh, the boundary 2D mesh must be completely enclosed.

  2. For convenience, midas FEA permits size control on various geometry types such as Solid, Face and Edge. Yet, the predominant factor for determining mesh size is what is given on the Edges. Therefore, in order to mesh with desired size properly, it is recommended that the mesh size on the critical edges be specified first.

  3. Once a lower dimensional mesh is created manually, a higher dimensional mesh can be generated automatically based on the lower dimensional mesh. *i.e. If a boundary 2D mesh is created, 3D mesh can be generated directly with it. Therefore, in theory, what is required to generate a 3D mesh is a boundary 2D mesh, not a Solid (Geometry).*

**Boundary mesh errors can occur more likely during 3D mesh generation.**

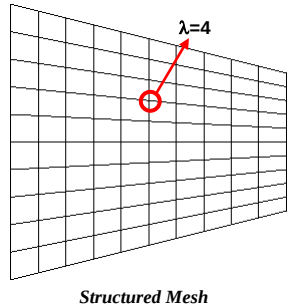
Valence is also known as “Degree of Regularity” or in short “Regularity.”

Valence is the number of elements, which share a single internal node. Depending on the Valence, the mesh can be divided into two groups, structured mesh and unstructured mesh.

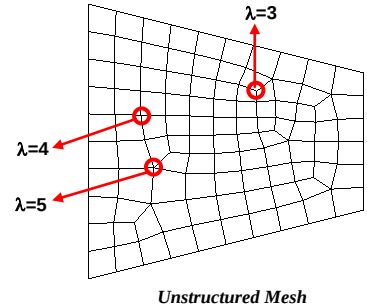
- **Structured Mesh**
  In the structured mesh, the Valence of internal nodes is constant.
  The ***Map Mesh*** function in midas FEA generates the structured mesh type in both 2D and 3D.

- **Unstructured Mesh**
  In the unstructured mesh, internal nodes have different Valences.
  The ***Auto Mesh*** function in midas FEA generates unstructured mesh with various auto-meshing algorithms.

The following figures will help us understand the definition of the Valence and the difference between the structured mesh and the unstructured mesh by using 2D mesh sets.




*Structured Mesh*




*Unstructured Mesh*

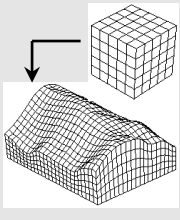
The Valence of 4 is ideal in generating 2D Quadrilateral Mesh (Structured Mesh). Whenever possible, the Valence between 3 and 5 is recommended.

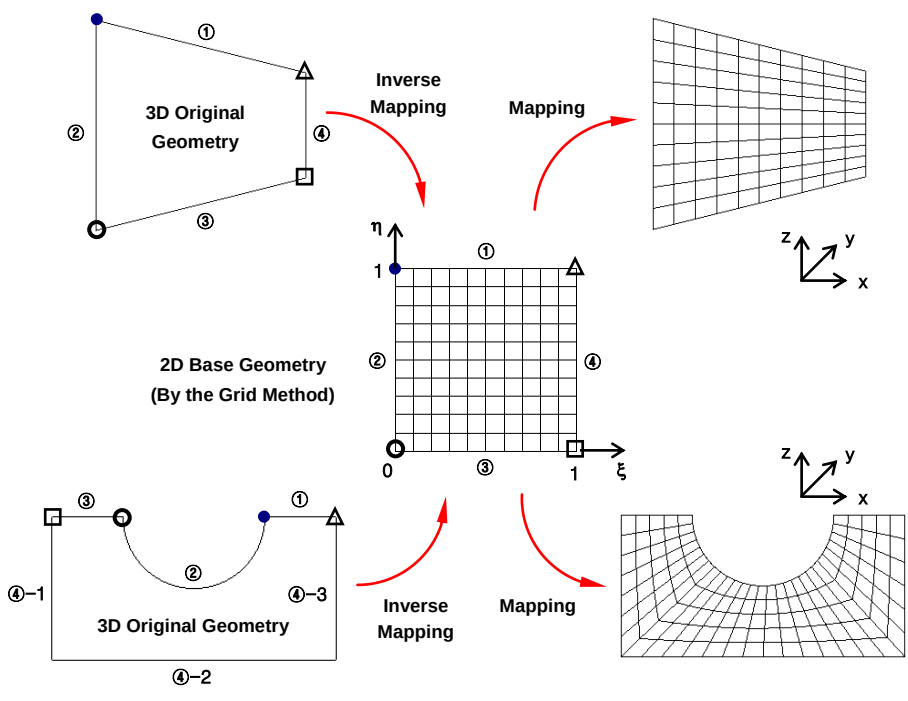
In the structured mesh of the above figure, all internal nodes have a Valence of 4, and each node is equally shared by 4 elements. Each element will have a corner angle of about 90° (360°/4). Such elements with close to a 90° corner angle result in excellent element quality. However, the internal nodes in the unstructured mesh may have different Valences such as 3 and 5, and the adjoining elements at each node will have a corner angle of about 120° and 72° (360°/λ) respectively. As the Valence deviates from 4, the element quality thus deteriorates.

Structured mesh renders excellent mesh quality, but there are a few trade-offs. The following diagram shows the general process of generating structured mesh, and the required conditions for generating structured mesh are indicated.

This type of structured mesh generation is known as "Mapped Mesh."

The shape of the base geometry of 2D structured mesh is rectangular. Similarly, for the 3D structured mesh, a cube becomes the shape of the base geometry.





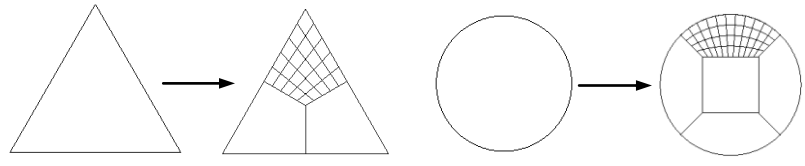

***Process of generating Structured Mesh***

After transforming the actual 3D geometry into the 2D base geometry, quadrilateral elements thus fill in the internal area of the base geometry. Again, the generated quadrilateral mesh is mapped back to the original geometry. Due to the complicated nature of this method, the original 3D geometry must satisfy the following two conditions.

**In a 3D structure, 8 corner vertices must be defined.**

1. In order to form the base geometry, 4 corner vertices must be defined.

   ➢ In case of convex polygons, midas FEA automatically detects the 4 corner vertices. Moreover, it also provides the user an option to define the 4 vertices manually. If 4 vertices cannot be defined, the geometry shape must be divided into applicable shapes as shown in the examples below.



2. The number of nodal divisions on an edge or an edge group must match the coupled edge or the coupled edge group (①-③, ②-④).
   It is recommended that the ***Number of Divisions*** be applied for the mesh size control rather than the ***Interval Length*** for constructing a structural mesh, in order to maintain the same number of mesh divisions on each side.

**Hence, unstructured mesh is referred to as Free Mesh.**

Unlike the structured mesh, the unstructured mesh can be applied to any shape of geometry without any limitations such as the number of divisions and the geometrical structure. Therefore, any complex geometry can be meshed with much less effort.

FEA provides the following 3 unstructured meshing algorithms:

- **Loop Mesher**

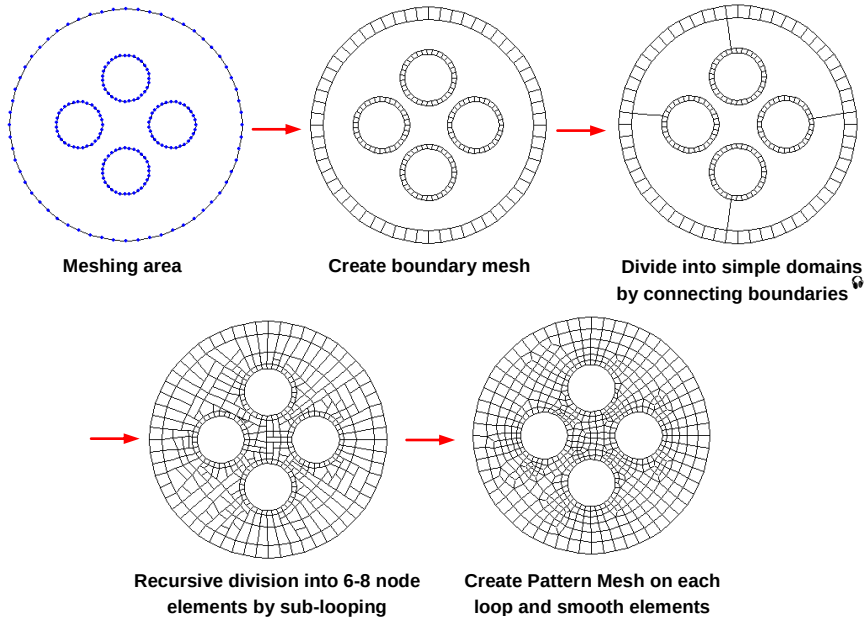
  Loop Mesher generates quadrilateral mesh on an object shape directly using a looping method. Loop Mesher is able to accurately encompass the outline of the object shape. And it generates uniform size mesh regardless of curvature. Especially, when the length to width ratio of mesh is more than 2, it generates quadrilateral mesh with a very smooth transition. However, due to the characteristics of the algorithm, the resulting mesh tends to become less regular and perpendicular. If an internal vertex or edge is included, the mesh quality may become relatively poor compared to other meshers.

- **Grid Mesher**

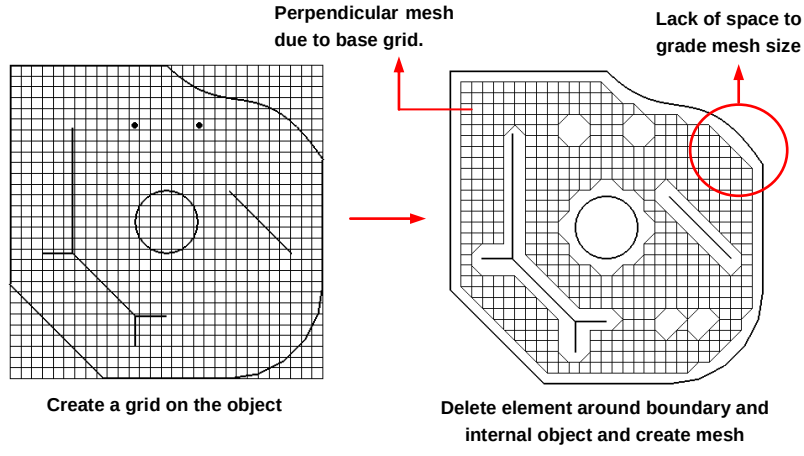
  Grid Mesher generates quadrilateral elements within a domain by using a base grid. And it connects internal mesh to boundaries using quadrilateral or triangular elements. This meshing algorithm is known to result in prominent quality in regularity and orthogonality. However, it exhibits a weakness when element size changes more than the ratio of 2 due to rough transitional characteristic between elements.

Grid Mesher can mesh with both quadrilateral and triangular mesh, and it is most suitable for domains that are likely to have similar mesh sizes.

**Looping Algorithm is only applicable to simply-connected areas.**




***Meshing process for Loop Mesher***




***Meshing process for Grid Mesher***

- **Delaunay Mesher**

  Delaunay Mesher is one of the most popular meshing algorithms. It exhibits excellent performance in triangular mesh generation. Especially, internal vertices and curves can be freely included in mesh generation.

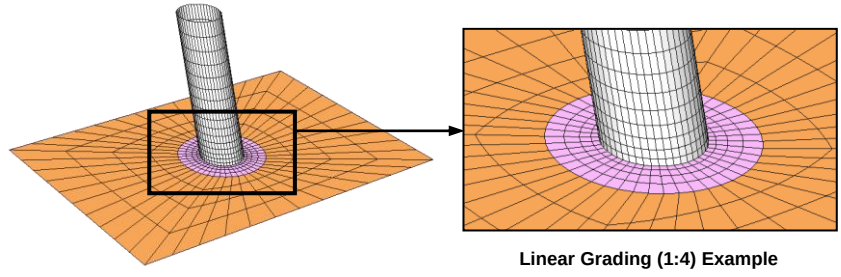
Proper selection of element types and the quality of element shapes affect the analysis results to a large extent. Therefore, it is very important to choose proper element types and shapes and check the nodal connectivity between the elements in order to obtain accurate results.

The following list introduces the essential information required before mesh generation:

1. As the element shapes become closer to regular polygons and regular polyhedrons, the analysis results become much more accurate.

2. The analysis time is a factor that should not be ignored during modeling. Therefore, it is important to keep the number of elements within a reasonable range. Careful planning of element distribution can prevent a massive model size. Fine mesh should be considered in the following cases;

(a) where the analysis is of a critical type, (b) where the geometry radically changes, (c) where the material or the property changes, (d) where the load varies, and (e) where critical results are expected. Coarse mesh can be then used for the balance area since the analysis results may not be very sensitive to mesh sizes.



**Linear Grading (1:4) Example**

**Example of Mesh where finer elements are generated near connection**
**(The selected element sizes should be such that a smooth transition is maintained)**

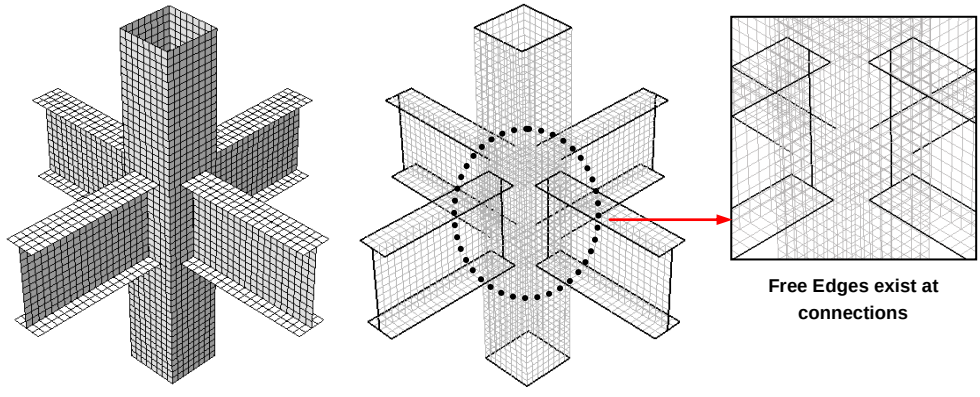
**However, good quality triangular or tetrahedral elements are better than poor quality quadrilateral or hexahedral elements.**

Since triangular elements are relatively stiffer than quadrilateral elements, a model with quadrilateral elements is generally expected to produce better results. Similarly, hexahedral elements are preferred to tetrahedral elements.

Also first order elements are stiffer than second order elements. Therefore, it is recommended that second order elements be used, especially when a model is assembled with triangular and tetrahedral elements.

**Free edge and free face can be checked by the "Check Mesh Function" of midas FEA**

3. Upon completion of mesh generation, the nodal connectivity between the generated elements must be checked. The separated nodes will cause free edges in 2D and free faces in 3D mesh.



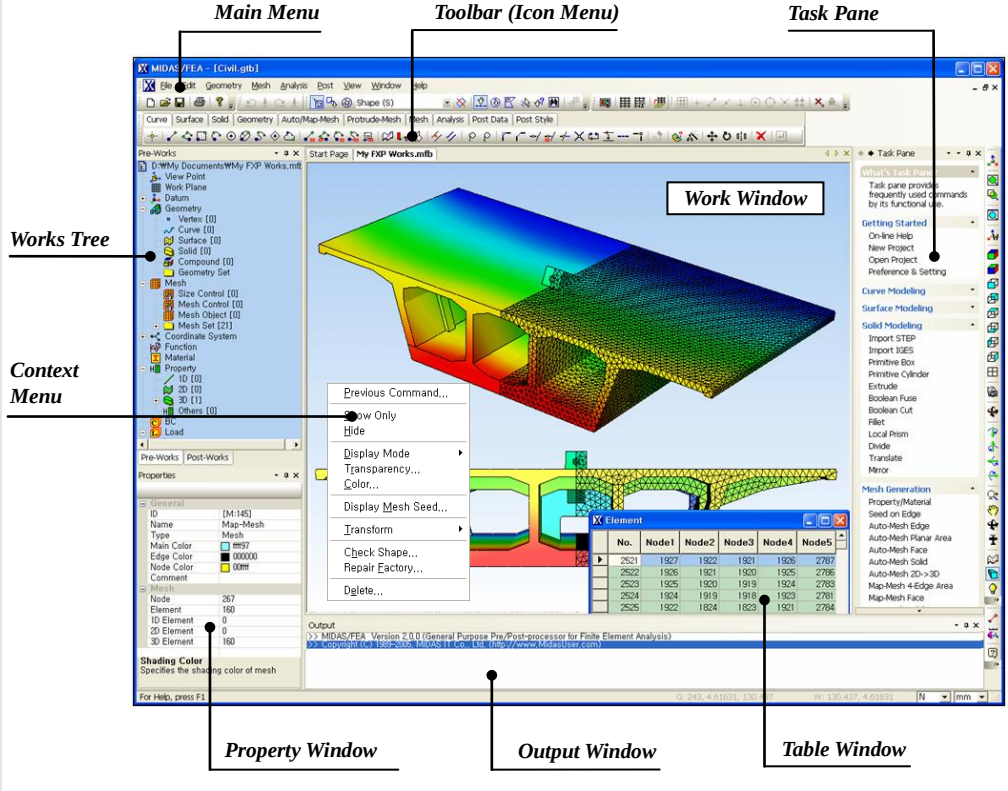

*Example of Free Edge in 2D Mesh at the Beam and Column Connection*

**In midas FEA, the user can check mesh quality with various criteria. Detailed information on the mesh quality function is available in the On-line Manual.**

4. The mesh quality checking tools in midas FEA enable us to verify the feasibility of mesh shape in an analysis model. The user must mesh carefully in the area where the geometry radically changes because unacceptable elements may occur. In addition, Jacobian Ratio for quadrilateral elements must be checked to see whether or not non-convex quadrilateral elements exist. In addition, collapsed tetras, nearly flat elements, must be avoided.

# 3.2 Organization of Window and Menu System

The Menu System of midas FEA permits an easy access to all the functions related to the entire process of input, output and analysis and minimizes mouse movements.




*Organization of Windows and Menu System*

midas FEA consists of a total of 6 windows.

- **Work Window**
  Work Window deals with modeling structures and interpreting analysis results by means of GUI (Graphic User Interface) of midas FEA.

- **Table Window**
  Table Window displays all types of data entries and analysis results in a Spread Sheet format. Various types of data modification, additional input, compilation, arrangement for different characteristics and searching capabilities are provided in Table Windows. The tables are compatible with common database software or MS-Excel for data transfers.

- **Works Tree Window**
  Works Tree stores all the data of the current project such as geometry, mesh, load, boundary condition, analysis control and analysis results. It organizes such data in a tree format, which is similar to Windows Explorer.

- **Property Window**
  Property Window shows general information about each individual entity that the user selects from the Work Window or the Works Tree. Basic modification such as its name and color can be also made through the Property Window.
  Depending on the working mode (pre-processing or post-processing), the Property Window has a unique format and function.

- **Output Window**
  Output Window displays all types of information necessary for modeling and warning and error messages.

- **Task Pane Window**
  Task Pane Window provides “Step by Step” guidelines, which allow easy access to most frequently used analysis options and modeling functions. midas FEA provides a general Task Pane definition, and the user is also free to customize it to his preference.

midas FEA has the following menu system:

- **Main Menu**

  The Main Menu has commands for all the built-in functions necessary to run midas FEA.

- **Toolbar (Icon Menu)**

  The Icon Menu helps the user promptly invoke functions frequently used in midas FEA. Each icon is regrouped with the icons of similar purposes by Tab for easier recognition.

**Curve Tab: The group of tools that are frequently used to create and modify curves**

Each Toolbar can be easily dragged with the mouse to a desired position on the screen. They may be edited to appear selectively on the screen or modified by using ***View>Toolbars>Customize***. For more information on any icon in the Toolbar, place the mouse cursor on the icon in question and Tool Tip will provide a short description.

- **Context Menu**

  In order to minimize the physical motions of the mouse, simply right click the mouse. midas FEA automatically brings a menu system, which offers related functions or frequently used functions reflecting the working circumstances of the user.

**It is better for the beginner of midas FEA to control the commands with the Main Menu to comprehend the program more thoroughly. However, once the user becomes familiar with the system, it is much more convenient to use the Icon Menu.**

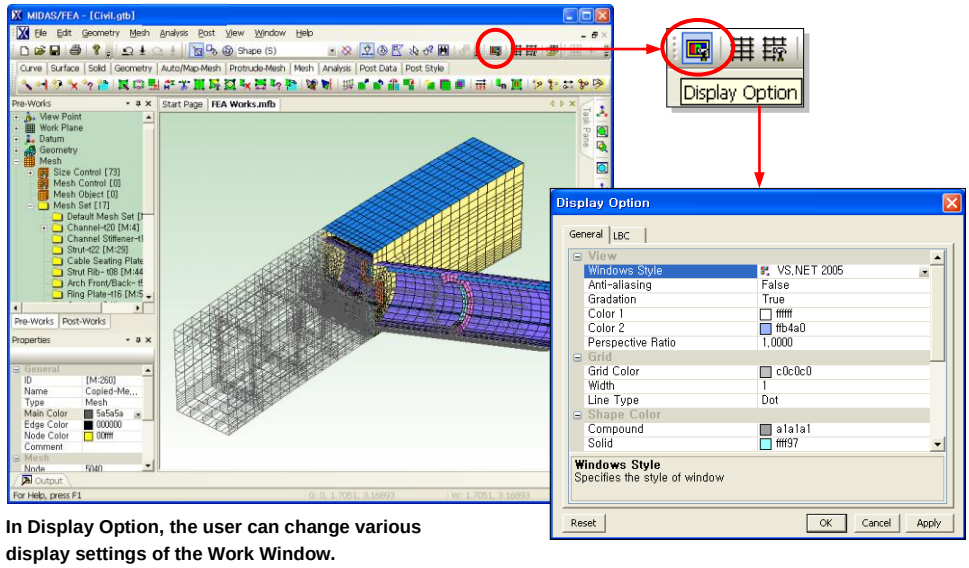
### 3.2.1 Work Window

When the user starts midas FEA, only the Start Page will appear on the screen. When opening a new project, the user will see the Work Window where actual modeling will take place.

The Start Page is a window that is hyperlinked to MIDAS IT Corporation's Homepage (***http://www.MidasUser.com***) and the User Support Center.

The Work Window deals with modeling and interpreting analysis results by means of GUI (Graphic User Interface) of midas FEA.

The display related options, including the modification of the background color, can be controlled in ***Main Menu > View > Display Option*** or ***Display Option*** icon in the Icon Menu.




**In Display Option, the user can change various display settings of the Work Window.**

The user can switch between the Start Page, Work Window and Table Window by clicking their Tabs.

### 3.2.2 Works Tree Window

midas FEA provides Pre-Works Tree and Post-Works Tree depending on working modes.

- **Pre-Works Tree**

  The Pre-Works Tree is similarly structured to the Windows Explorer. It retains all pre-processing modeling information of geometry, mesh, load and boundary conditions. The user can also perform basic modeling operations using the Pre-Works Tree such as Show/Hide objects.

  The following are the featured functions of the Pre-Works Tree:

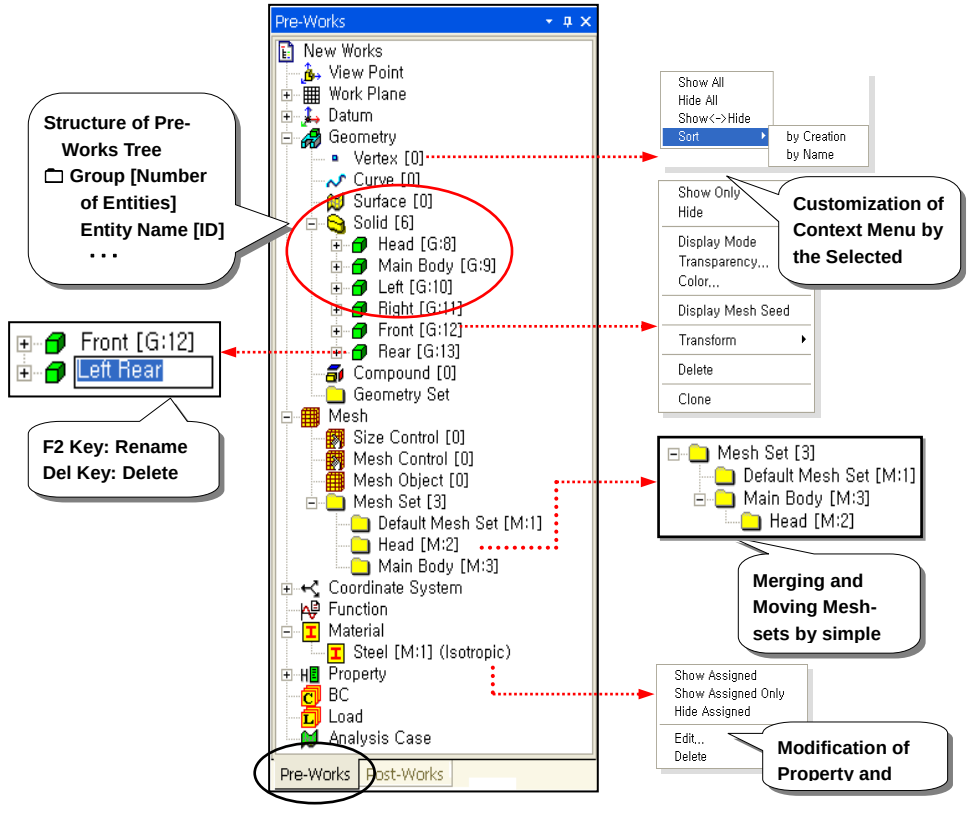
  1. The user can select objects from the Pre-Works Tree as well as from the Work Window.

  2. Depending on the selected entity, the Pre-Works Tree provides a customized Context Menu, which includes a range of practical operations such as the Show/Hide function.

  3. The Works Tree provides a working environment similar to the Microsoft Windows Explorer with which the user is familiar. The user will likely operate various functions intuitively such as Rename, Delete and Move in the Works Tree.

**When dealing with a complex model, the user often encounters troubles in selecting and modifying entities from the Work Window. However, using Works Tree and Context Menu can help the user select and modify the entities much more easily and conveniently. Since Context Menu changes depending on the selected entity, the user is encouraged to try out each case individually.**

**The user can learn more about its functions in the provided Tutorials.**

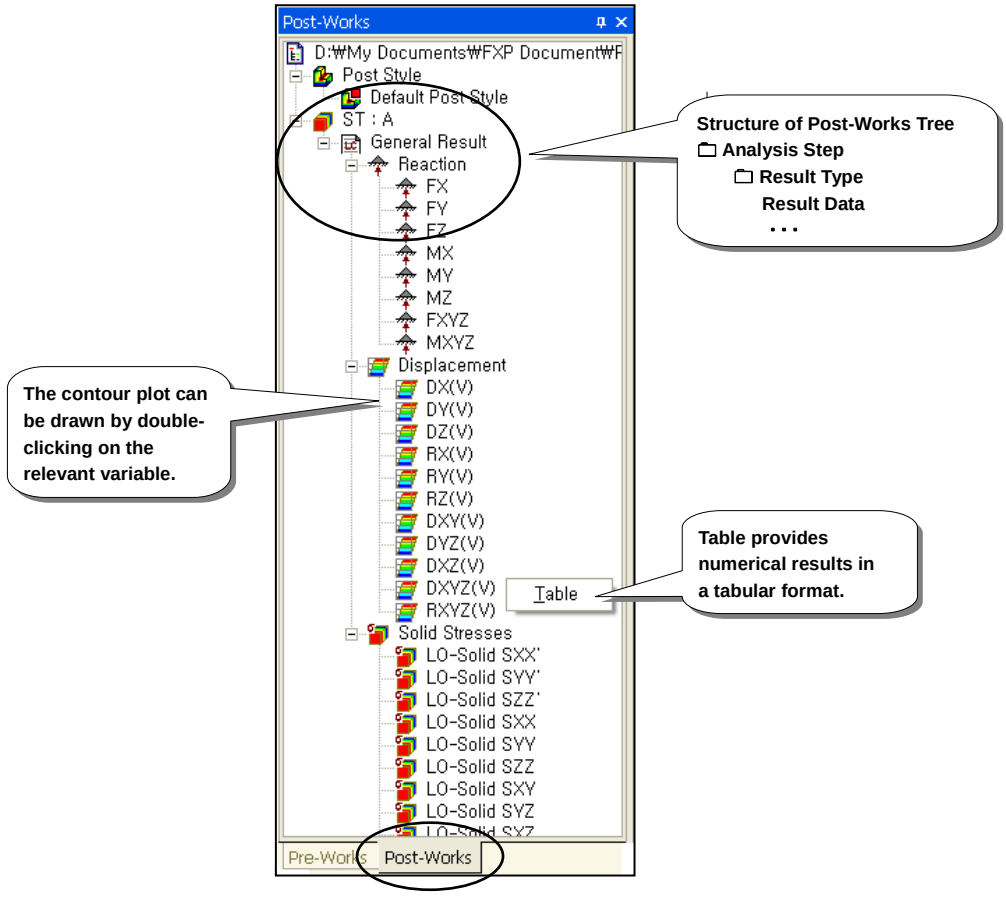



***Structure of Pre-Works Tree and Its Functions***

In other programs, the user must check the analysis results after careful interpretation. However, midas FEA organizes results in a tree format automatically, and the user can simply view the post data by simple mouse clicks.

- **Post-Works Tree**

  Once the analysis is completed, the program automatically brings the post data and arranges it in a tree format. In the Post-Works Tree, the user can control most of the output results such as the graphic and table related displays. The user can also save the current display output.




***Structure of Post-Works Tree and Its Functions***

### 3.2.3 Property Window

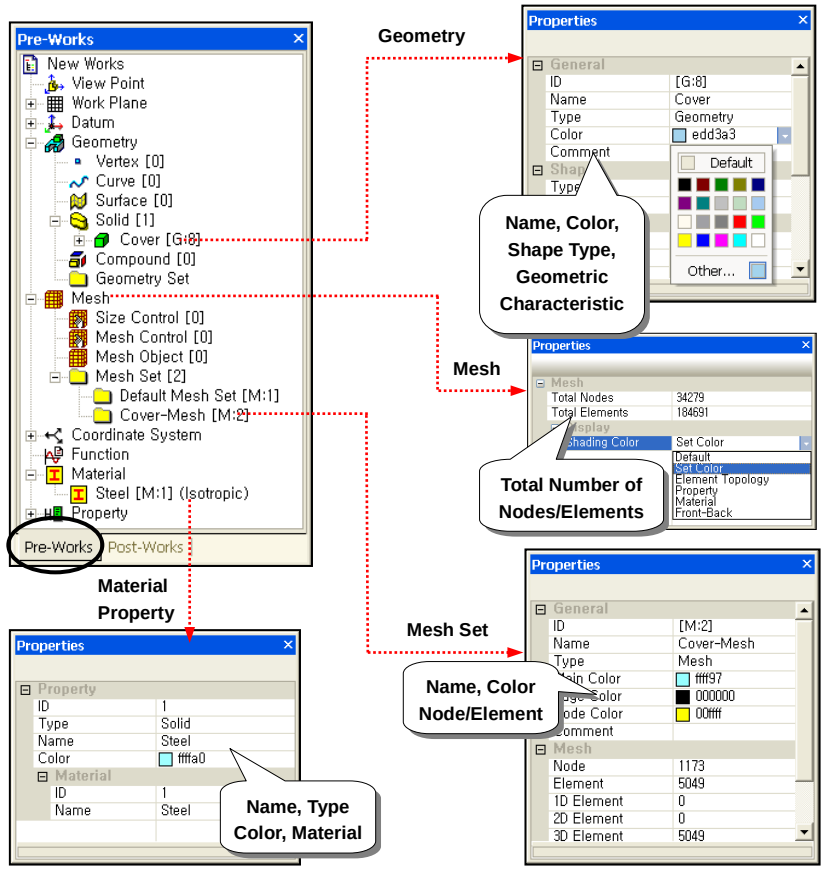
The Property Window provides a set of information about the object that has been selected from the Work Window or the Works Tree. Depending on the actual working mode, the Property Window functions differently. Therefore, the Property Window will have a different structure for each mode (Pre-Processing and Post-Processing).

- **Pre-Processing Mode**

  The Pre-Processing Mode displays general information on a selected object, and simple modification of basic properties such as name and color can be made in the Property Window. The structure of the Pre-Processing Property Window may change depending on the selected entity, but the general structure is shown in the following figure:

**The selection's name and color can be changed in the Property Window.**

In order to invoke the Property Window, only one single object must be selected.



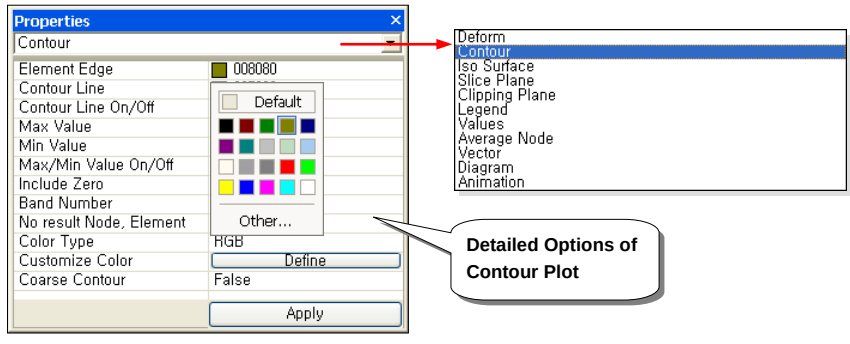

***Example of displayed Information in Pre-Processing Property Window***

Post-options are specified in the Property Window, and post-operations are controlled in the Icon Menu.

- **Post-Processing Mode**

  Unlike the information shown in the Pre-Processing Mode, the Property Window in the Post-Processing Mode is used to specify options for the post-processing operations.

  Once an option is specified in the Property Window, it becomes applicable to all post processing operations. In addition, the program saves it into the registry so that the same option can be applied to other projects.

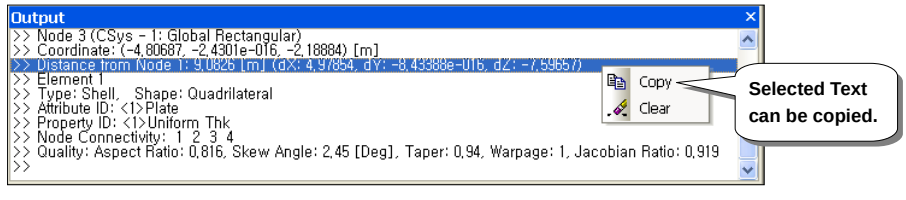



***Structure of the Property Window in the Post-Processing Mode***

### 3.2.4 Output Window

The Output Window displays all types of information necessary for modeling and analysis, warning and error messages.

The displayed messages in the Output Window can be copied to the Clip Board, and they can be used in other text programs.



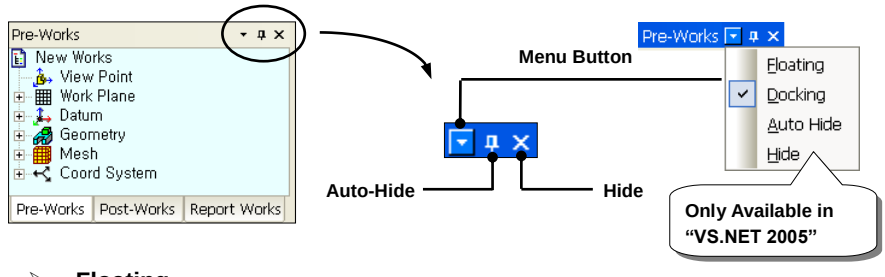

***Example of Messages in Output Window and its Context Menu***

## Operations of Works Tree/Property/Output Windows

**The menu button is available only in the VS.NET 2005 format. The window style can be specified in Display Option (*View > Display Option*). Floating and Docking, which do not require specific buttons, can be controlled by dragging the mouse.**

While working with the Works Tree Window, the Property Window and the Output Window, there may be a need to hide or change their locations.

Here, the basic operation of these 3 windows will be introduced. All the three windows have the main control buttons on the top right corner. Basically, all the operators are included in the Menu Button as shown in the figure, and for convenience, midas FEA provides additional individual buttons for most frequently used functions.



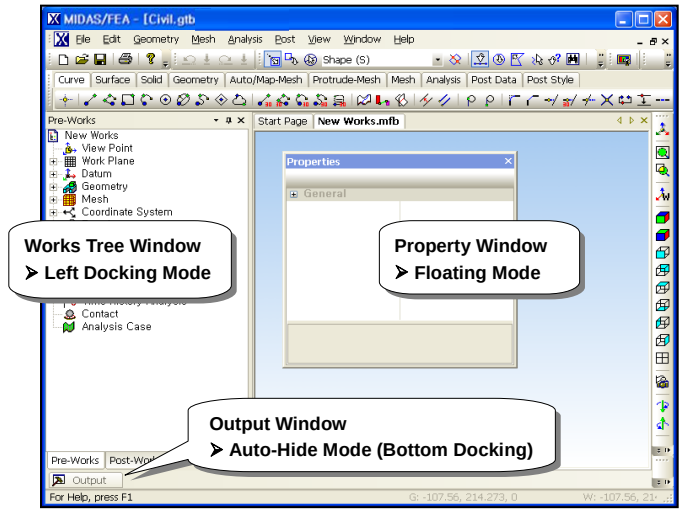

**Auto-Hide can be used only in the Docking mode.**

- **Floating**
  The window can be located at any place in and out of the program.

- **Docking**
  The window can be located only at a designated location such as Right/Left/ Top/Bottom.

- **Auto-Hide**
  The window will be hidden except for the title bar. When placing the mouse over the title bar, the entire window will appear again.

- **Hide**
  The window will be completely hidden. The window's Show/Hide is controlled through the Window menu in the Main Menu.




To move the window location, hold down the left mouse button after clicking on the title bar and release it at a desired location.

- **To position a window in the Floating mode**
  After dragging a window into an empty space, release the left mouse button.

- **To position a window in the Docking mode**
  Dragging a window into the boundaries of the main frame or the inside of another window will set the window to the Docking mode. When the window is positioned in a desired docking location, release the mouse button to complete the operation.
  The easiest way to situate a window properly in the Docking mode is:

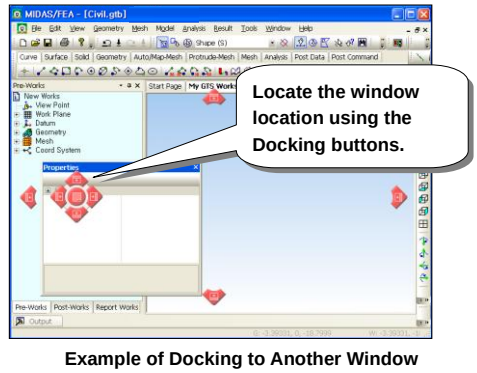
1. Specify the window style as the ***VS.NET 2005*** format in the General tab of **Display Option** (***View > Display Option***).

**The default window style is the VS.NET**

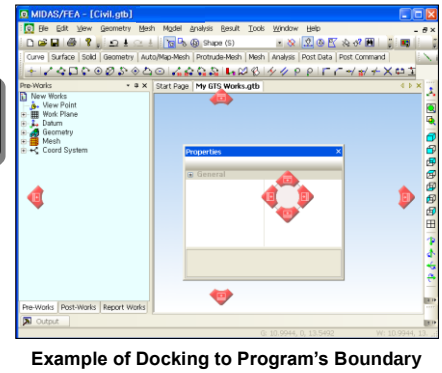
2. Click the title bar of the object window, and drag the mouse around. The docking position buttons will appear on the screen. The buttons will be structured differently depending on where it is dragged.

3. Place the mouse pointer on the top of the desired docking position button. When the position is highlighted, release the mouse button to place the window in the Docking position.

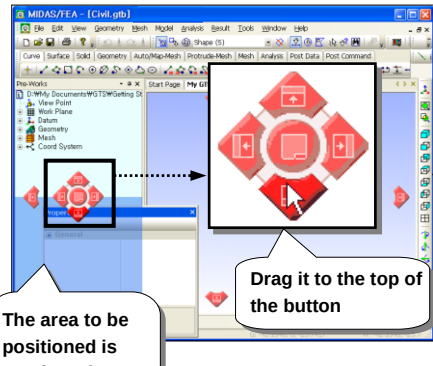
This method is very convenient when the user is trying to dock a window inside another window.

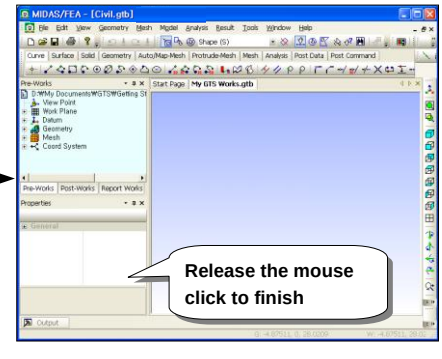



**Example of Docking to Another Window**
**(Drag the positioning window to another window)**



**Example of Docking to Program's Boundary**
**(Drag the positioning window to Work Window)**



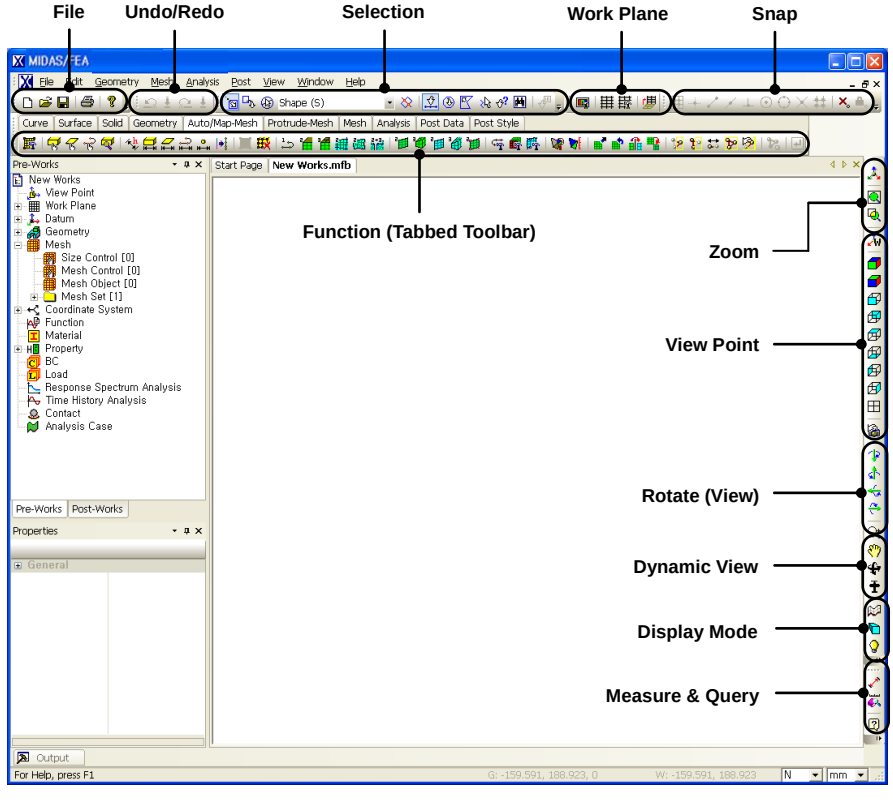
### 3.2.5 Main Menu

Main Menu includes commands and shortcut keys of all the functions that are required to operate midas FEA.

| | |
|---|---|
| ***File*** | File, Print and Data Exchange |
| ***Edit*** | Undo/Redo |
| ***Geometry*** | Geometry Modeling Functions (Create/Modify), Work Plane and Grid |
| ***Mesh*** | Mesh Control / Mesh Generation (Auto, Map, Protrude) / Manual Operation |
| ***Analysis*** | Analysis Option / Analysis Control / Analysis Perform |
| ***Post*** | Result Combination / Extraction / Post File Conversion |
| ***View*** | Display Option / Show and Hide / Visualization Related |
| ***Window*** | Window's Active / Inactive / Arrangement |
| ***Help*** | On-line Help / Link to MIDAS IT Homepage / E-mail Contact |

### 3.2.6 Toolbar and Icon Menu

Icon Menu helps the user promptly invoke functions, which are frequently used in midas FEA. Each icon is regrouped with the icons of similar purposes in various Toolbars. Each Toolbar can be easily dragged with the mouse to a desired position on the screen. For further information on any icon in the Toolbar, place the mouse cursor on the icon in question and Tool Tip will provide a short description.



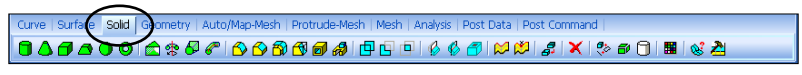

***Configuration and Arrangement of Icon Menu***

The Function Toolbar is one of the outstanding features in midas FEA. The Function Toolbar is designed in the tabbed toolbar format to maximize its usability. Therefore, even a novice can utilize the toolbar immediately. The commands are grouped together by their geometrical or functional types. For example, if the user wishes to make a planar surface, he can find relevant commands in the Surface Tab.

Example: How to find an icon in the Function Toolbar?

**Function Toolbar is most widely used Tool bar. Therefore, it is recommended that each icon be tried.**

| | | |
|---|---|---|
| To Create a polyline (which is a geometric type of Curve.) | → | Curve Tab |
| To Trim an object (which is a geometric type of Solid.) | → | Solid Tab |
| To delete a polyline or a solid | → | Geometry Tab (a group of commonly shared functions by multiple geometry types) |




**After completing Solid operations in the Solid Tab in Icon Menu,**

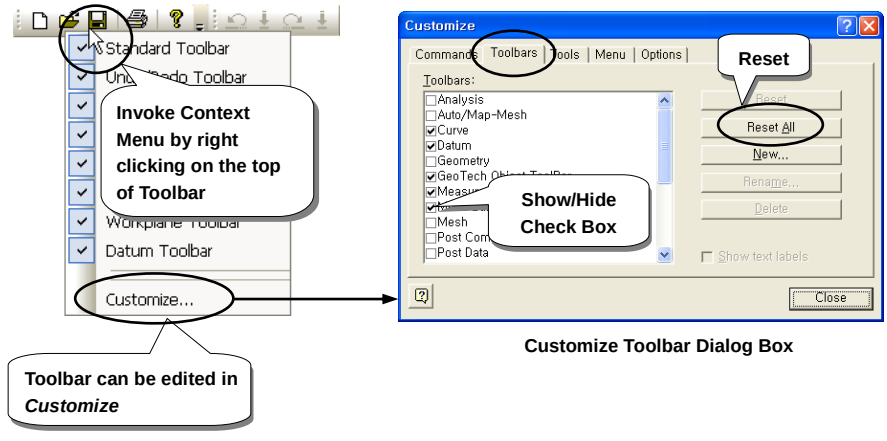
**Switch to the Auto/Map-Mesh Tab, and Generate Mesh on the Solid.**

***Example of Effective Use of the Function Toolbar***

The detailed information on the Toolbars and the Icon Menu of midas FEA can be found in the Appendix, "**Toolbar & Icon Menu.**"

**In the Command Tab of the Customize Toolbar dialog box, the user can edit the Toolbars by dragging icons in and out. The customized Toolbar can be initialized with Reset All button in the Toolbars Tab.**

The Toolbars can be placed in any desired area of the program, and they can be shown or hidden through Customize command in the Context Menu. In addition, the user can customize the Icon Menu so that they can create their own Icon Menu.




**Customize Toolbar Dialog Box**

***Customization of Toolbar***

## 3.2.7 Context Menu

In order to minimize the physical motions of the mouse, simply right click the mouse. midas FEA automatically selects a menu system, which offers related functions or frequently used functions reflecting the working circumstances of the user.

Please refer to the Appendix for the detailed information on midas FEA's Context Menu.

# 3.3 Selection and Manipulation of Work Window

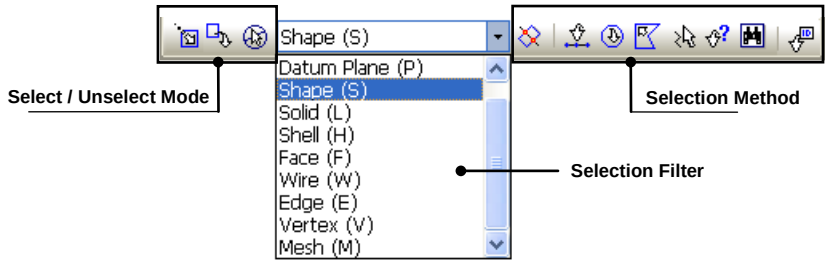
midas FEA offers various Selection methods and Work Window handling capabilities to effectively generate a model.

## 3.3.1 Selection

**In order to understand all the selection functions, the user must become familiar with the geometry concept thoroughly. If you are unfamiliar with this concept yet, please review Chapter 2.1.2 Modeling Process of midas FEA.**

The ***Selection*** functions are extremely important and indispensable for the overall task of generating a model. midas FEA provides various selection methods so that the user can handle any complex model effectively in any situation.

All selection methods are controlled from Selection Toolbar.




***Organization of midas FEA's Selection Toolbar***

The parenthesis indicates the shortcut key for each selection method. This works only when the focus is on the Work Window.

| | | |
|---|---|---|
| | ***Select (0)*** | Add to selection. |
| | ***Unselect (0)*** | Remove from selection. |
| | ***Unselect All*** | Unselect all the selected entities. |
| | ***Include Intersected (Ctrl)*** | Select all the entities that are contained inside the selecting area as well as the entities intersecting the defined boundaries. |
| | ***Pick / Window (1)*** | Select entities by clicking the mouse once each time. Or select the entities in a rectangular bounding area defined by dragging the mouse cursor. |
| | ***Circle (2)*** | Select all the entities in a circular bounding area defined by dragging the mouse cursor. |
| | ***Polygon (3)*** | Select entities by successively clicking the corners of a polygon containing the relevant entities with the mouse cursor. |
| | ***Polyline (4)*** | Select entities by crossing a series of lines that intersect the desired entities with the mouse cursor. |
| | ***Query Pick (5)*** | Select entity from the list of the entities, which are detected at the mouse click on the Work Window. |
| | ***Displayed*** | Select all the entities, which are currently shown in the Work Window. |
| | ***ID*** | Select the node(s) or the element(s) by entering its Node ID or Element ID. |

When the user brings the mouse pointer above a desired entity, it will become highlighted with a sky blue edge. Therefore, the user can check in advance whether or not the selection will be made correctly. Once the entity is selected, the boundary color will be changed to pink.

- **Select**

  It invokes the Select Mode to select entities.

  The user can switch to the unselect mode by typing “0” on the keyboard or by pressing the middle button of the mouse.

- **Unselect**

  It invokes the Unselect Mode to unselect the selected entities.

  The user can switch to the select mode by typing “0” on the keyboard or by pressing the middle button of the mouse.

- **Unselect All**

  This unselects all the selected entities.

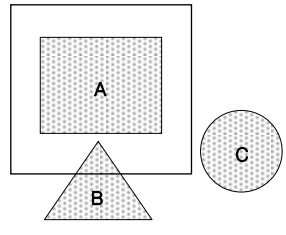
  Double clicking the mouse scroll button also executes the function.

**Use caution while operating Unselect All. Depending on the speed of double clicking the mouse, the user may inadvertently switch between the select / unselect modes.**

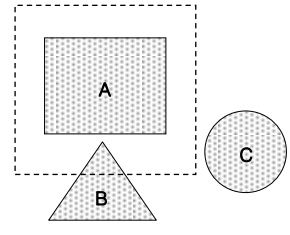
- **Include Intersected**

  When entities are selected with ***Window***, ***Circle*** or ***Polygon,*** it selects all the entities that are contained inside the selection area, as well as the entities intersecting the boundaries of the area.

**While operating the Include Intersected Mode, the selection boundary is drawn in dotted lines.**




**Window Selection (Include Intersected: Off)**
**➢ Selection to be made: Only A**




**Window Selection (Include Intersected: On)**
**➢ Selection to be made: A & B**

The user can toggle the Include Intersected Mode on and off by pressing the **Ctrl** key while dragging the selection area.

- **Pick / Window**

  Using the Pick Select, desired entities can be selected by clicking the mouse once each time. The Window Select feature can be affected by dragging the mouse from one corner to the other.

  - **Pick Select**

    Select desired entities by clicking the mouse once each time. To unselect the selected entities, click them once again in the Select Mode.

**The direction of dragging a selection window is not important in midas FEA. The inclusion of the entities intersected by the window boundary is controlled by the Include Intersected Mode.**

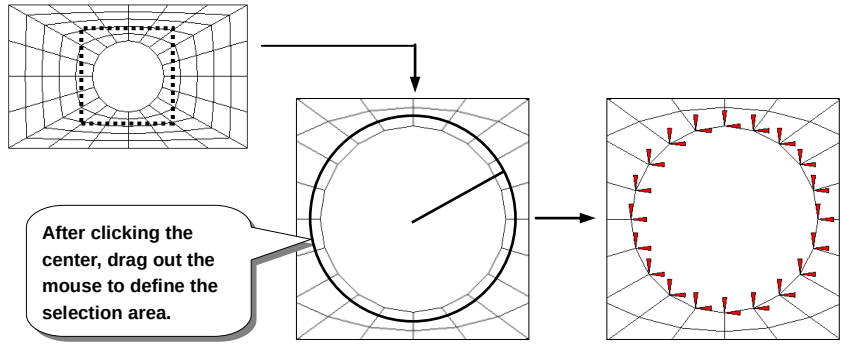
➢ **Window Select**

Click the diagonal corners of a window area containing desired entities with the mouse cursor and select or unselect the desired entities completely contained within the rectangular window. Press **ESC** to cancel while defining the window area.

Enter the ***"1"*** key to switch to the Pick/Window Selection method during the modeling process.

- **Circle**

Select desired entities that are contained in a circular bounding area.
The circular area is defined by clicking the center point and dragging out to a desired radius. Press ***ESC*** to cancel while defining the circular area.

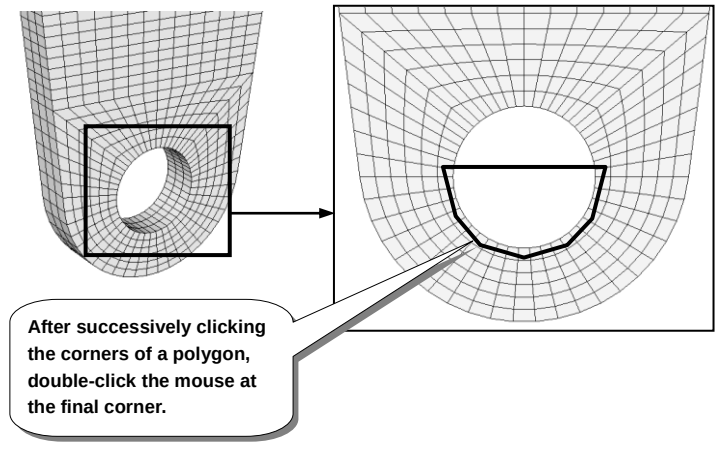



***Example of Applying a Boundary Condition using Circle Selection***

Enter the ***"2"*** key to switch to the Circle Selection method during the modeling process.

- **Polygon**

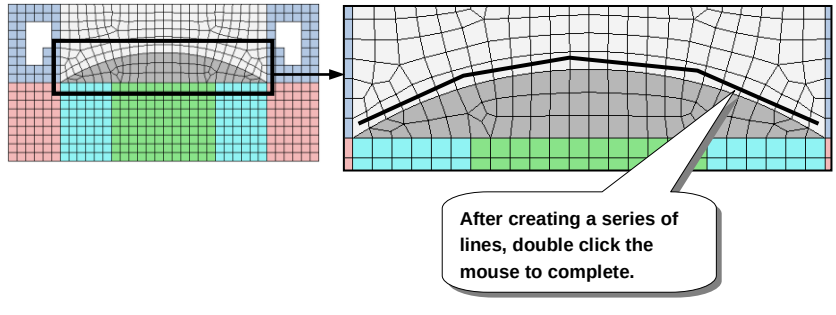
**S**elect or unselect desired entities by successively clicking the corners of a polygon containing the relevant entities with the mouse cursor. When clicking the final corner, double-click the mouse to end. Press ***ESC*** to cancel while defining the polygon.




Enter the ***"3"*** key to switch to the Polygon selection method during the modeling process.

- **Polyline**

  Select or unselect entities by drawing a series of lines containing the desired entities with the mouse cursor in the Model Window. When clicking the final point of the last line, double click the mouse to end. Press ***ESC*** to cancel while defining the polyline.

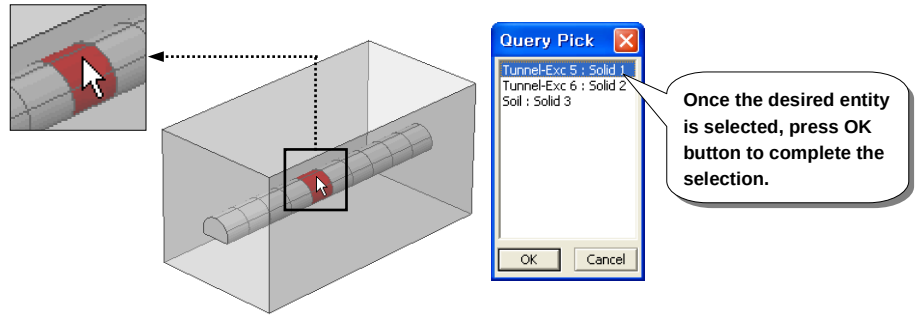



***Example of Selecting Faces Using the Polyline Selection***

Enter the ***"4"*** key to switch to the Polyline selection method during the modeling process.

- **Query Pick**

  Select or unselect desired entities from the list of all the entities, which are detected at the mouse click location. Going down the list, each entity on the list will be highlighted. Click OK for the desired entity to select.




*Example of Selecting a Solid Using the Query Pick Selection*

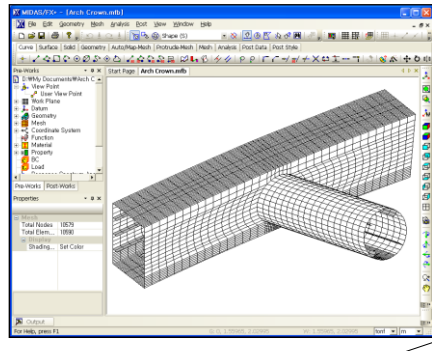
Enter the "5" key to switch to the Query Pick Selection method during the modeling process.

- **Displayed**

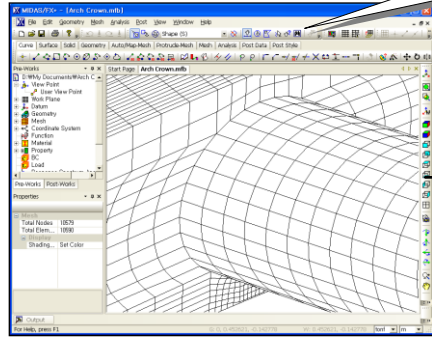
  Select or unselect all the entities shown in the Work Window.

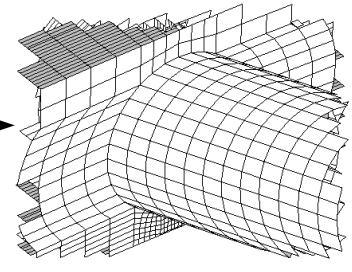
  Simply click the ***Displayed*** icon from the Selection Toolbar.

**When Work Window is set to Zoom All, the Displayed selection will work identically as the Select All command.**



*Example of the Displayed Selection at "Zoom All" view*



- **ID**

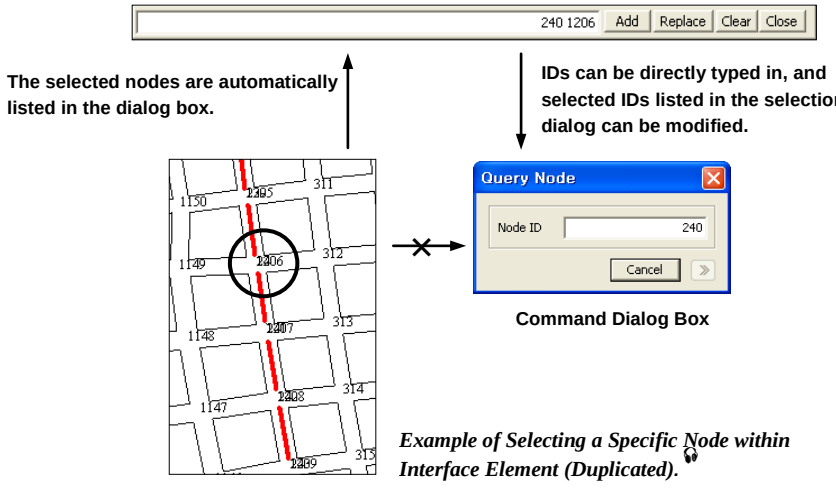
In the node and element selection mode, the user directly enters the Node or Element ID(s) to select desired entities. The user can distinguish the Node and Element Selection Mode by Selection Filter. This mode is only active for the node and element manipulation functions. When the ID selection method is on, the ID Selection dialog box appears on the Work Window. The detailed selection procedure is explained below.

**The Selection Filter will be explained in the next section in detail.**

1. Enter the ID(s) of either Node or Element in the Dialog Box to make a selection.

2. If nodes and elements are selected by different selection methods, the ID(s) of the selected entities will appear on the dialog box. The user can modify the list of the selection to complete.

**The ID Selection Dialog box behaves like an intermediary between the selection manager and the Command Dialog.**




*Example of Selecting a Specific Node within Interface Element (Duplicated).*

The ID selection is convenient for selecting specific elements when multiple elements are located at the same location.

The ID Selection dialog box consists of the following buttons.

| | |
|---|---|
| ***Add*** | Add the newly entered node(s) or element(s) ID(s) to the selection. |
| ***Replace*** | Unselect the existing selection, and select the newly entered ID(s) for nodes or elements. |
| ***Clear*** | Clear the list in the ID selection dialog box. |
| ***Close*** | Close the ID selection dialog box. |

Even after the user clears the Selection Dialog Box, the previously selected entities still remain selected.

The Selection Filter enables the user to select only by the specified entity type. Depending on the working mode and the active command, the Selection Filter changes its form, and the user can choose the entity type within the specially customized Selection Filter. The working mode in terms of the Selection Filter is basically distinguished by whether or not a command is invoked.

- **Neutral Mode**

The Datum represents reference points, lines, and planes.

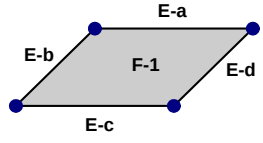
The neutral mode is when the user has not called a command yet with the Selection Filter set to the default. In the neutral mode, the user only can select Datum, Shapes and Mesh Sets. The sub-Shape of a geometric entity and the individual nodes and elements cannot be selected.

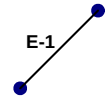
- It is recommended that the Selection Filter be set to the Shape in the Neutral Mode.

- The Selection Filter changes its form by the relevant command. Ensure to check the Selection Filter before proceeding.

- **Command Mode**

  The Command mode is activated when the user has called a specific command, and the Selection Filter has formed with the relevant type of the function. The user can select any desired type including the sub-Shape and the individual node and element depending on the type of the command.




| Shape | Type |
|---|---|
| F-1 | Face (4 Sub-edges) |
| E-1 | Edge |

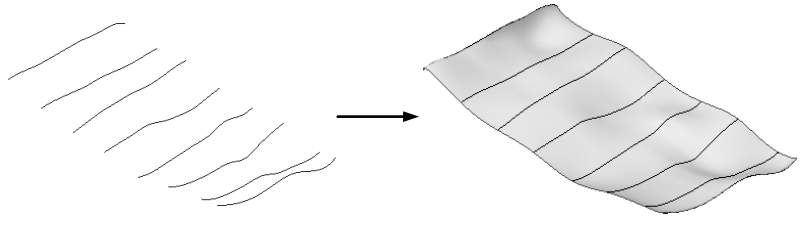
| Selection Filter | Working Mode | Selected Entity |
|---|---|---|
| **Shape** | Neutral Mode | 2 (F-1, E-1) |
| | Command Mode | 2 (F-1, E-1) |
| **Edge** | Neutral Mode | 1 (E-1) |
| | Command Mode | 5 (E-1, E-a, E-b, E-c, E-d) |

***Difference in selected entities between the Neutral Mode and the Command Mode***

**Why is there such a difference in selection between the working modes?**

Although the same type of entity has been specified in the Selection Filter, the selected entities turn out differently depending on the current working mode. The difference occurs because only the independent geometric entity (Shape) is allowed in the selection in the Neutral Mode. This concept may seem complicated at first, but it becomes highly beneficial when it comes down to advanced modeling applications.

The following example explains where the selection restriction in the Neutral Mode can be effectively used. A 3-dimensional face is created by connecting 8 curved edges.



**Shape (8 Edges)**

**Shape (1 Face, 8 Edges)**
**Edge (Total 12, 8 edges for Shape and 4 edges for Face)**

Since the original eight edges are no longer needed, they should be either deleted or hidden. Let us carefully look into either case, when the Sub-shape selection is restricted or when it is allowed.

- **If all geometric entities including sub-Shapes are allowed in the selection,**

  1. **With "Shape" in the Selection Filter**
     A total of 9 Shapes (1 Face and 8 Edges) are selected.

  2. **With "Edge" in the Selection Filter**
     A total of 12 Edges are selected. 4 of them belong to the Face, and they cannot be separately treated by the program.

**Each edge can be picked manually one by one, but it still causes difficulties since some edges are duplicated at the same location.**

**In actual modeling, such an operation can be also performed easily by the use of the Works Tree.**

**Some commands restrict the sub-Shape in selection due to a similar reason as in this example.**

- **If only the Shape is allowed in the selection,**
  By having the Selection Filter for Edge, only the 8 unnecessary edges will be selected, which can be deleted or hidden all together.

In order to perform such an operation easily, midas FEA allows only the Shape to be selected in the Neutral Mode.

In the Command Mode where modeling processes are mostly carried out, both the Shape and the Sub-shape can be used freely. Therefore, in actual operation, the selection difference between the Neutral Mode and the Command Mode does not create any problems, and in fact it helps during complex modeling.

The general components of the Selection Filter are shown in the following table.

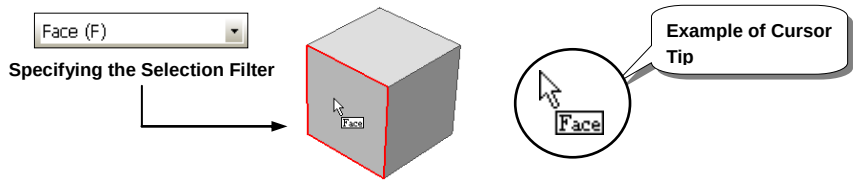
**In the table, the letters in the parentheses represent the shortcut keys for the Selection Filter components. The shortcut keys work only when the focus is on the WorK Window.**

| Selection Filter | Subjected Entity |
|---|---|
| **Datum Axis (A)** | Select Datum Axis.<br>➢ To define direction (Translate/Extrude/Project), revolution axis, etc. |
| **Datum Plane (P)** | Select Datum Plane.<br>➢ To define the plane to project, to divide, and to mirror. |
| **Geometry** <1> | |
| **Shape (S)** | Select Shape. |
| **Solid (L)** | Select Solid. |
| **Shell (H)** | Select Shell. |
| **Face (F)** | Select Face. |
| **Wire (W)** | Select Wire. |
| **Edge (E)** | Select Edge. |
| **Vertex (V)** | Select Vertex. |
| **Mesh** | |
| **Mesh (M)** | Select Mesh Set.<br>➢ To select Mesh Set itself or to select nodes and/or elements that are included in the Mesh Set. |
| **Node** <2> | Select Node. |
| **Element** <2> | Select Element. |
| **Node/Element** <3> | Select Node and Element at the same time.<br>➢ To include or exclude nodes and elements in the Mesh Set. |
| **Element-Face** <4> | Select a 3D element face.<br>➢ To apply Face Pressure Load on a 3D element face. |
| **Element-Edge** <4> | Select a 2D element edge.<br>➢ To apply Edge Pressure Load on a 2D element edge. |

<1> The sub-Shape of geometric entities will not be selected in the Neutral Mode.
<2> Only applicable for the Node/Element modifying functions.
<3> Only applicable for the "**Incl/Excl**" function in the Mesh Set Menu.
<4> Only applicable for the Load and Boundary Condition functions.

The user can control whether or not to display the type at the Cursor Tip in the Display Option (*View > Display Option*).

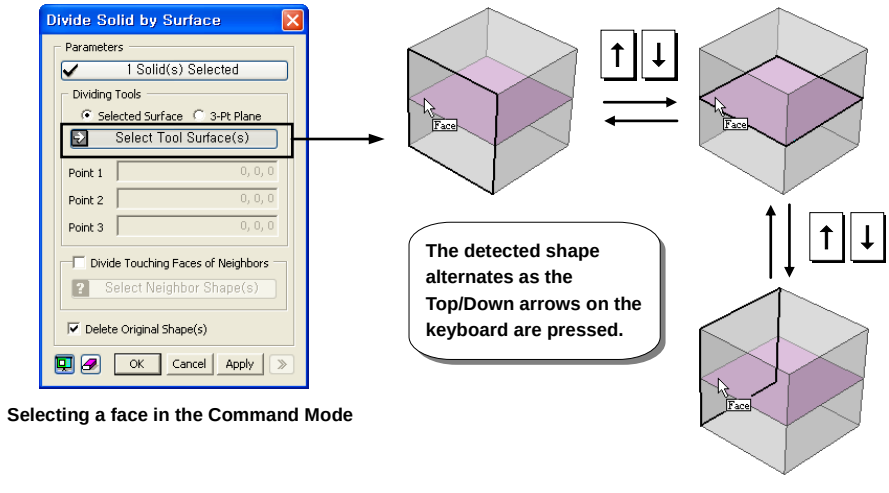
The specified type in the Selection Filter will be displayed at the Cursor Tip on the screen.




Since all the sub-Shapes can be selected in the Command Mode, the number of selectable entities increases tremendously for lower level entities. Therefore, it is very difficult to select a proper entity among many duplicated items by using the mouse cursor. In order to alleviate such a problem, midas FEA provides the following functions:

This method works only in the Command Mode.

1. When the selection method is on "**Pick**", the highlighted entity at the mouse cursor can be alternated by pressing the Top/Down arrows of the keyboard.




**Selecting a face in the Command Mode**

2. The Query Pick selection can help the user to properly select a desired entity.
   The user can carefully check the selection from the list of the entities, which are detected at the mouse position.

**Most problems are associated with the Select / Unselect Mode or the Selection Filter.**

When any function related to the selection does not operate properly, check the following items for troubleshooting:

1. Check whether or not the selection mode is the Select Mode or the Unselect Mode.

2. Ensure that the entity type is correctly specified in the Selection Filter.

3. Try again at a different mouse position.

**If the selection sensitivity is set very high, too many objects can be detected. This tolerance also affects the snap tolerance. Therefore, use caution when changing the selection sensitivity.**

4. Change the selection sensitivity (allowable tolerance).
   The selection sensitivity can be modified in ***File > Preference*** Menu.

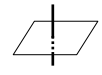
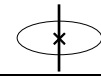
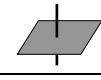
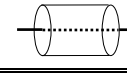
Most cases will be resolved by (1)~(3), and (4) must be used only in special situations.

The geometric characteristic and topology can be also implemented in the modeling process in midas FEA. Therefore, the geometric entities, which include special characteristics such as direction and plane can be selected.

The following examples are most frequently used cases for utilizing the geometric characteristic and topology in midas FEA.

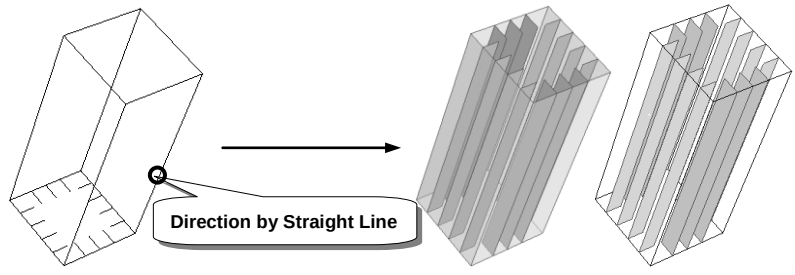
- **Define Direction or Revolution Axis**
  When executing commands such as Translate, Extrude, Project, Rotate and Revolve, the user must define a direction or revolution axis to complete the operation. In midas FEA, the direction and the revolution axis can be defined by using various entities, which relate to relevant characteristics.

  The entities that can be used to define directions and revolution axes are listed in the following table:

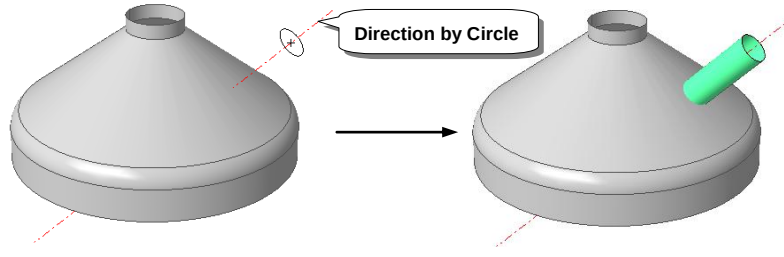
| Allowed Entity | Geometric Characteristic | Selection Filter |
|---|---|---|
| **Datum Axis** <1> | The direction of Datum Axis | ***Datum Axis*** |
| **Datum Plane** | The normal direction of Datum Plane  | ***Datum Plane*** |
| **Edge (Straight Line)** | The direction of a straight line | ***Edge*** |
| **Edge (Circle/Arc)** | The normal direction of a plane defined by a circle or an arc.  | ***Edge*** |
| **Face (Plane Face)** | The normal direction of a plane face  | ***Face*** |
| **Face (Revolved Face)** | The central axis of a revolved face (cylinder, etc)  | ***Face*** |
| **2 Point Vector** | A user-defined 2 point vector | N/A |
| **Normal of Profile** <2> | The direction normal to an executing object (planar entity) | N/A |

<1> The Datum Axis is set as default in the Selection Filter for defining the direction and the revolution axis.

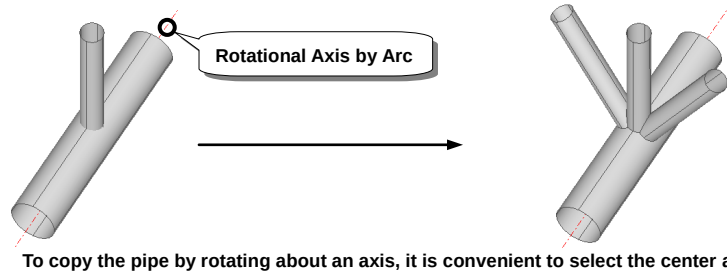
<2> Its use is limited to some special functions such as Extrude and Local Prism.




**To create stiffeners in 3D space, 2D edges are extruded in the direction of an existing straight line.**




**To create a pipe from a circle, the extrusion direction can be defined by the normal direction of a plane where the circle is located.**




**To copy the pipe by rotating about an axis, it is convenient to select the center arc as the revolution axis. (The revolved body face can be also selected.)**

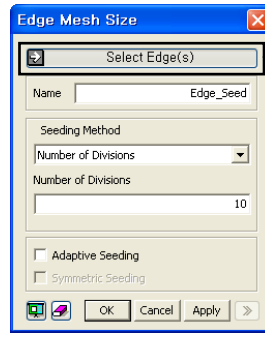
As shown in the above examples, any direction and rotational axis can be easily defined by using the unique characteristic of a geometric entity. In addition, midas FEA provides various snap functions to define a 2 point vector and a 3 point plane, which can be very effective during the process of complex modeling.

| Grid | Vertex | End | Middle | Ortho | Center | Quadrant | Intersection | Node | All-Off |
|---|---|---|---|---|---|---|---|---|---|

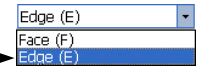
***The Snap Toolbar of midas FEA***

- **Select by Means of Topological Relationship of Geometry Shape**

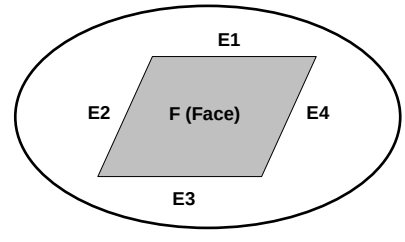
  Due to the topological concept in midas FEA, sub-Shapes can be selected by using higher level geometry. In other words, by selecting a higher level entity, the user can select all of its sub-Shapes at once.




**Mesh Size Control along Edge (Ready to Select Edges)**




**Structure of Selection Filter**

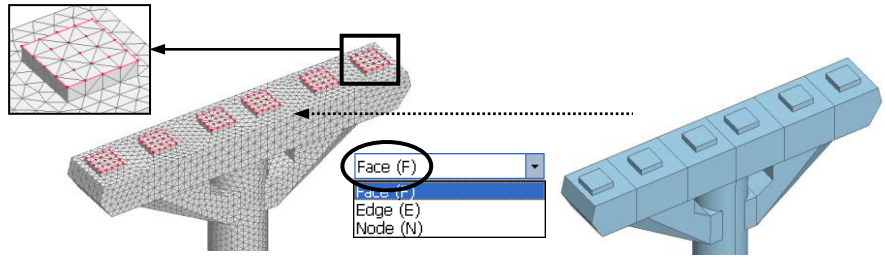



For example, in the above case, each edge will be selected separately if the Selection Filter is set to Edge. However, by setting the Selection Filter to Face, the user can select the four sub-Edges at once. The use of a higher level entity is thus recommended to select all of its sub-Shapes.

- **Select Node and Element**

  Similar to the geometry selection, after setting the Selection Filter to Mesh, the user can select Mesh Sets from the Work Window or the Works Tree. Since nodes and elements are included in a Mesh Set, all of them will be selected together at the same time. If mesh is generated on the geometry, its nodes and elements can be selected by the geometry unit such as Face and Edge.

**For selection by Geometry, it must be shown on the screen.**

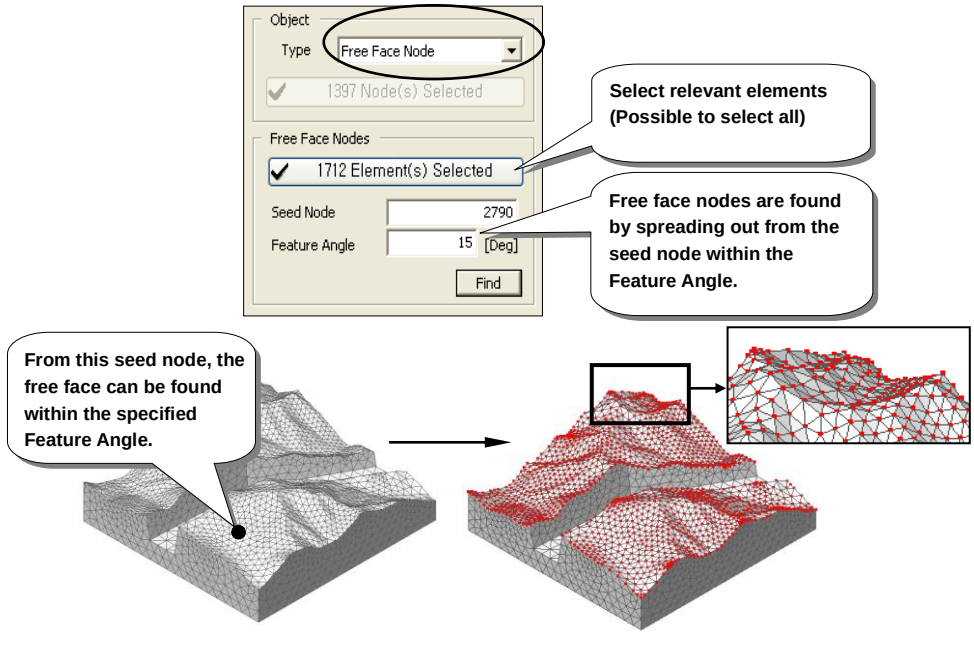



*Example of Selecting Nodes which belong to 6 Top Faces*

- **Selecting Nodes or Element-Faces Using Feature Angle**

  For maximum work efficiency in applying load and boundary conditions, midas FEA offers additional selecting tools, which are customized especially for defining load and boundary conditions. The most frequently used tools among them are the Free Face selection in 3D mesh and the Free Edge selection in 2D mesh.

**This selection method does not use geometry relationships. Therefore, it is useful when mesh is generated manually without geometric shapes.**




*Example of Selecting Top Face Nodes (Free Face Node in 3D Mesh)*

- **Other Useful Selection Tools**

  When selecting nodes and elements, the user can choose a specific material or property from Pre-Works Tree. It detects all the nodes or elements to which the selected material or property is assigned.

  In a similar way, selecting a load or boundary condition from Pre-Works Tree will make the selection of relevant nodes and elements as well. This feature is very useful when modifying or deleting previously assigned load and boundary conditions.

### 3.3.2 View Manipulation Functions

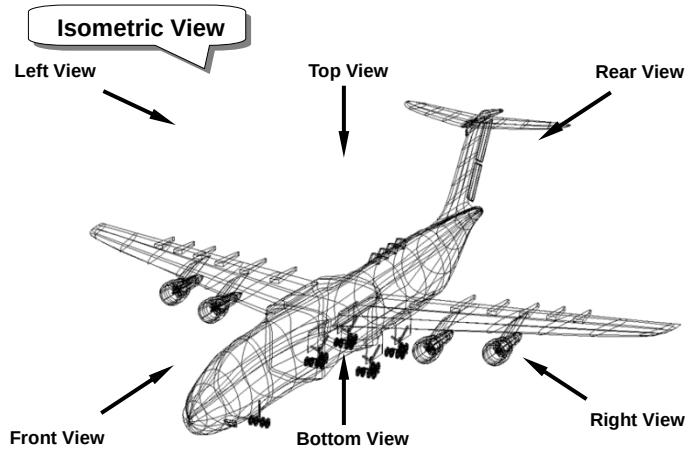
All the ***View Manipulation*** functions of midas FEA assist the user to accurately grasp the three-dimensional views of the model input state and the analysis and results through diverse view angles and points.

Most of the View Manipulation functions are available in Icon Menu of the View Toolbar.

***The Organization of the View Toolbar***

| | | |
|---|---|---|
| | ***Reset View*** | Initialize the viewpoint and the screen size to default. |
| | ***Zoom All*** | Fit the model to the screen size by scaling up/down. |
| | ***Zoom Window*** | Assign a desired size of the window by dragging a corner of the window with the mouse. |
| | ***Iso View*** | Display the model in a three-dimensional space. |
| | ***Front View*** | Display the model as viewed from the –Y direction. |
| | ***Rear View*** | Display the model as viewed from the +Y direction. |
| | ***Top View*** | Display the model as viewed from the +Z direction. |
| | ***Bottom View*** | Display the model as viewed from the –Z direction. |
| | ***Left View*** | Display the model as viewed from the –X direction. |
| | ***Right View*** | Display the model as viewed from the +X direction. |
| | ***Normal View*** | Display the model as viewed from the positive normal direction of the Work Plane.<br>(The +X-axis of WP is displayed on the right, and –Y-axis is displayed at the top.) |
| | ***Rotate Left*** | Rotate the model to the left (clockwise about Z-axis). |
| | ***Rotate Right*** | Rotate the model to the right (counterclockwise about Z-axis). |
| | ***Rotate Up*** | Rotate the model upward from the horizontal plane. |
| | ***Rotate Down*** | Rotate the model downward from the horizontal plane. |
| | ***Dynamic Zoom*** | By dragging the mouse, zoom in/out the model view. |
| | ***Dynamic Pan*** | By dragging the mouse, move the model view. |
| | ***Dynamic Rotate*** | By dragging the mouse, rotate the model view. |

**The rotational angle at each click of the Icon Menu can be modified in the Display Option. (*View > Display Option*)**




The most frequently used view control functions are the Dynamic View Manipulations.

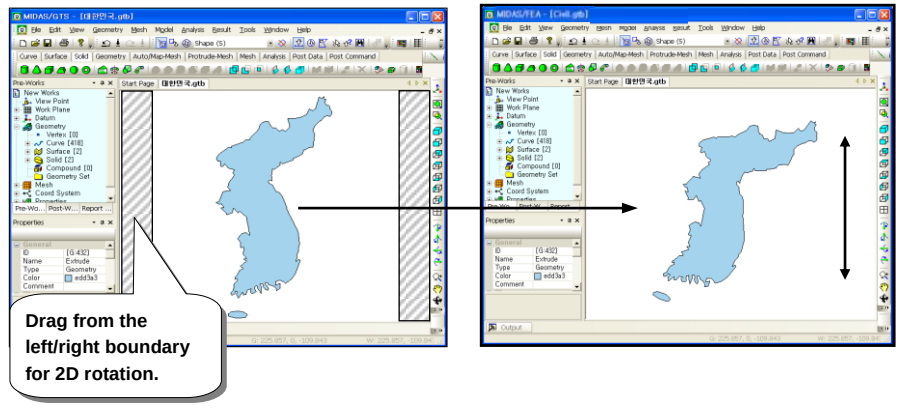
- **Dynamic Zoom ( , Ctrl+LB)**
  After clicking the Dynamic Zoom Icon, drag the mouse in the Work Window to zoom in/out the model view. Pressing the left button of the mouse and dragging the mouse downward or to the left reduces the window, and dragging the mouse upward or to the right magnifies the window. To continuously zoom in/out, drag the mouse around while pressing down both the “Ctrl” key and the mouse left button.

- **Dynamic Pan ( , Ctrl+MB)**
  After clicking the Dynamic Pan Icon, drag the mouse in the Work Window, and the model will follow the course of the mouse. To continuously pan, drag the mouse around while pressing down both the “Ctrl” key and the mouse middle button.

- **Dynamic Rotate ( , Ctrl+RB)**
  After clicking the Dynamic Rotate Icon, drag the mouse in the Work Window, and the mouse direction will rotate the model. To continuously rotate, drag the mouse around while pressing down both the “Ctrl” key and the mouse right button.

**It should be within a 10% range of the Work Window boundaries.**

The Dynamic Rotate behaves differently depending on the location where the mouse drag starts. If the user drags the mouse from the left/right boundaries of the Work Window, the model will rotate 2-dimensionally on the current view plane.




***Dynamic Rotate has different functionality depending on the dragging position.***

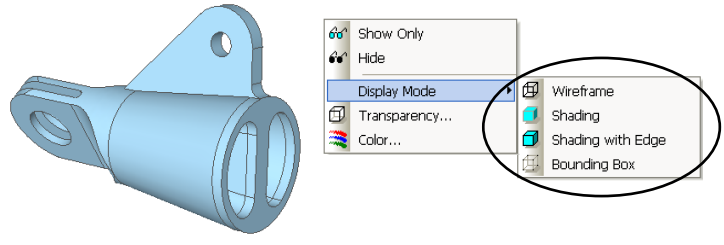
### 3.3.3 Model Representation

The Model Representation functions of midas FEA present the model in diverse displays and views. These functions help the user grasp the input state of the model and manipulate the model freely as desired.

- **Display Mode of Geometry**

  The display mode of geometry can be specified to an individual shape. To modify the graphical display, the user selects the desired shape(s) first and sets the Display Mode Menu, which can be found in the Context Menu.

**The object entity must be "Shape", so that multiple entities can be modified together.**




*Changing the Display Mode in the Context Menu*

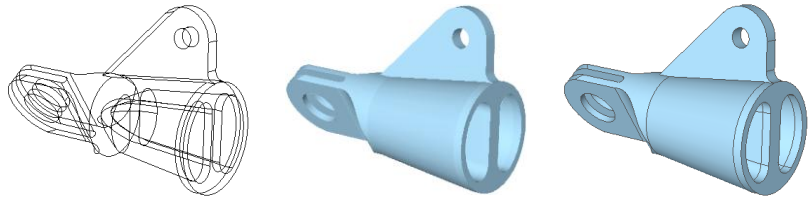
A geometric model can be displayed in 4 different modes.

1. **Wireframe**
   It only displays the boundary edges of the shape.

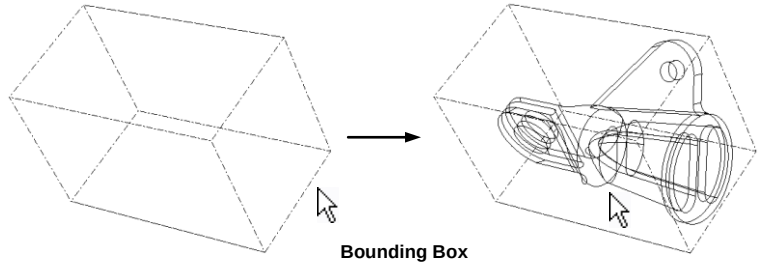
2. **Shading**
   It only displays the exterior faces of the shape.

3. **Shading with Edge**
   It displays both the boundary edges and the exterior faces.

4. **Bounding Box**
   Instead of displaying the actual shape, it displays a bounding box. When the user moves the mouse cursor into the bounding box, the geometric entity will be shown in the **Wireframe** Display.

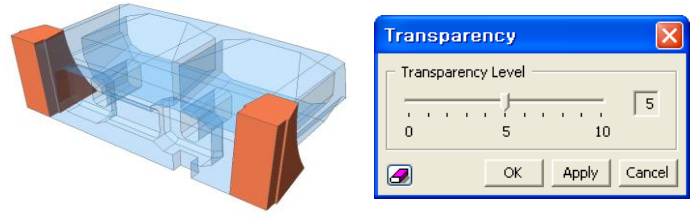


Wireframe Shading Shading with Edge

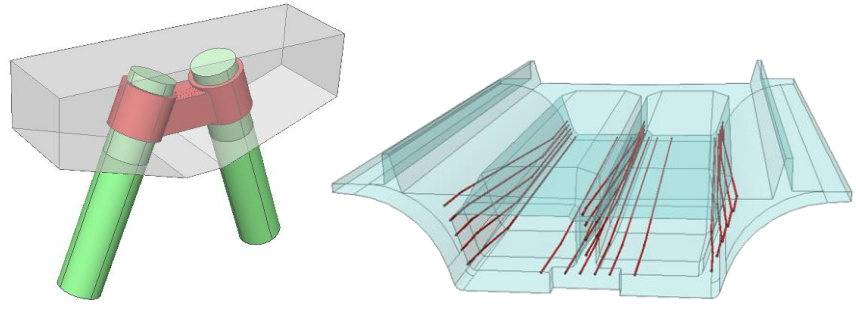


Bounding Box

In addition, midas FEA provides the **Transparency** option in the ***Shading*** or ***Shading with Edge*** display thereby allowing the user to see through the interior of the geometric shape. The Transparency option can be found in the Context Menu once the object shape is selected. In the Transparency dialog box, the transparency level can be adjusted. To remove the Transparency from the shape, set it to ***0*** or click the **Reset** button.




*Example of Transparency Display*



*Example showing the Interior Solids and Tendons by Transparency*

The Display Mode can be applied differently to each individual Shape. Therefore, it can be effectively used to model complex models, which have numerous geometrical shapes requiring different display identities.

- **Display Mode of Mesh**

  The display mode of mesh can be specified to each individual Mesh Set. Similar to the display mode of geometry, the user must select a desired Mesh Set before selecting the Display Mode menu in the Context Menu.

  The Mesh Set can be displayed in the following three different ways:

  1. **Wireframe**

     It hides the faces of all the elements in the Mesh Set and only displays the element edges.

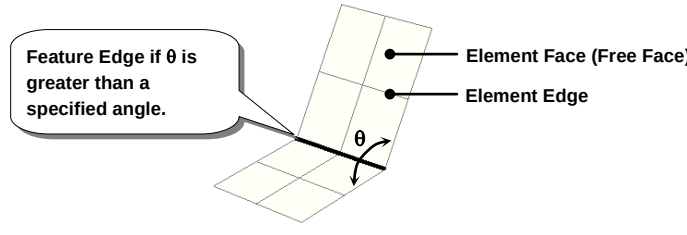
**2. Shading**

It shows both the edges and faces of the elements in the Mesh Set.

**3. Feature Edge**

It displays a Mesh Set with the Feature Edge. The Feature Edge represents the edge where the angle between the two connected free faces is greater than the specified angle. In this mode, the user can visually verify the shapes of internally hidden Mesh Sets.

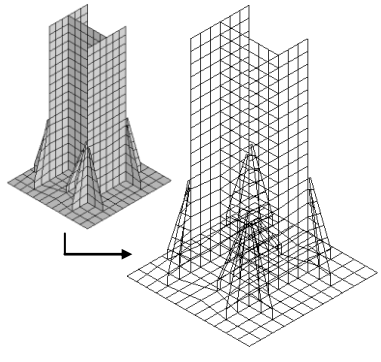
Its default angle is 30 degrees, and it can be adjusted in the Display Option (*View > Display Option*).



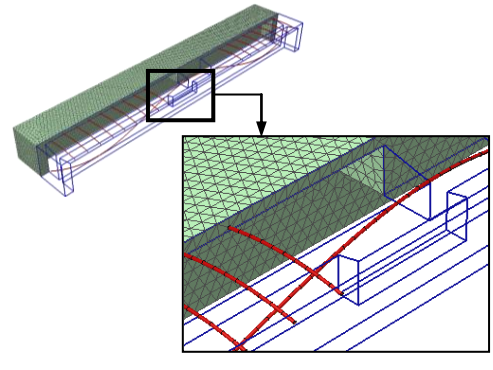

The default display mode of Mesh Set can be set in the Display Option (*View > Display Option)*.

**4. Free-Face Wireframe**

It shows the free faces of a Mesh Set in Wireframe. All internal elements will be hidden.



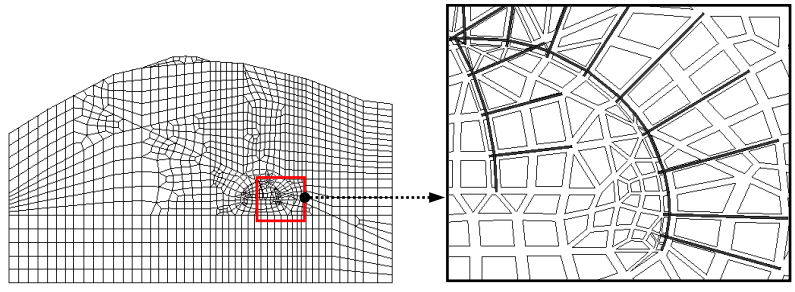
**Example of Wireframe Display**



**Example showing Tendon Locations by Displaying Solid Mesh by Feature Edge**

Similar to the graphic display capabilities of geometry, the display option can be applied to individual mesh sets.

The user can apply the Shrink option to each Mesh Set to display its elements in proportionally reduced sizes.



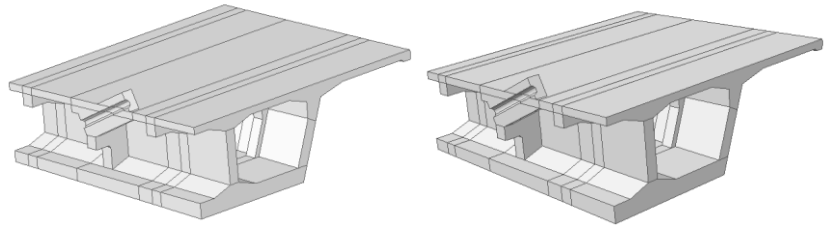
*Example of "Shrink" Display of a 2D Mesh Set*

- **Perspective Mode**

  The Perspective Mode displays the model in a perspective 3-dimensional view.

**The Perspective Ratio can be adjusted in the Display Option (*View > Display Option*).**

**In the Perspective Mode, the Grid Snap cannot be used.**



**Parallel Mode** **Perspective Mode**

### 3.3.4 Data Input Command

For convenience, midas FEA provides the following data entry options:

1. Where several numerical data are entered consecutively in a data field, these data can be distinguished by a "," (Comma) or a " " (Blank space).
   <Example> '333, 102, 101' or '333 102 101'

2. Where the same length is repeated, the entry can be simplified by "*number of repetition* @ *length*" instead of repeating the same number.
   <Example> 20, 25, 22.3, 22.3, 22.3, 22.3, 22.3, 88 → 20, 25, 5@22.3, 88

3. The keyboard may be used to enter selected data directly. The related node numbering or element numbering may be an arithmetic progression in series or the progression may be incremental. Then, the data entry can be simplified by "*start number* **to** (**t**) *final number*" or "*start number* **to** (**t**) *final number* **by** *increment*"
   < Example> 21, 22, … , 54, 55, 56 → "21 to 56", "21 t 56"
   < Example> 35, 40, 45, 50, 55, 60 → "35 to 60 by 5", "35 t 60 by 5"

4. Numbers and mathematical expressions can be used in combination. The majority of the operators and parentheses applied in engineering computation can be used.
   <Example> $\pi \times 20$ → PI * 20
   <Example> $35+3\times\left(\sin 30^\circ+2\sqrt{\cos 30^\circ+\sin 30^\circ}\right)$
   → "35 + 3 * (sin(30) + 2 * SQRT(cos(30)+sin(30)))"

Built-in operators in midas FEA are as follows:

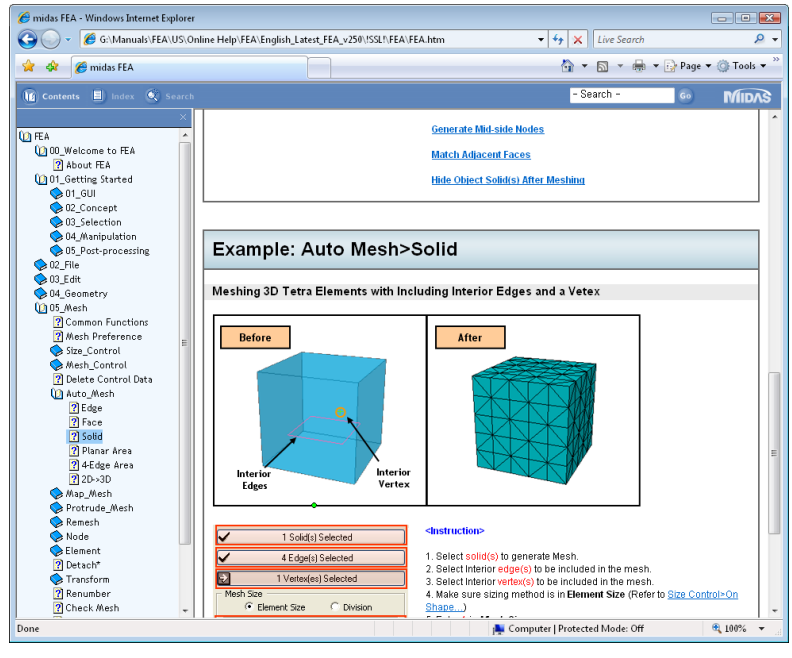
| Notation | Content | Remarks |
|---|---|---|
| ( | Open parenthesis | – |
| ) | Close parenthesis | – |
| + | Addition | – |
| - | Subtraction | – |
| * | Multiplication | – |
| / | Division | – |
| PI | $\pi$ | 3.141592653589793 |
| SQRT | $\sqrt{}$ | Ex.: $\sqrt{2}$ = SQRT(2) |
| SIN | Sine | Unit: 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 function | Ex.: $e^{0.3}$ =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 | – |

- Operators accept the mixed use of capital and lowercase letters.
- As the operators are similar to that of an engineering calculator, the hierarchy of operations follows the rules of common mathematical operations.

### 3.3.5 How to Use the On-line Manual

When using midas FEA, pressing F1 key or clicking the Help menu allows access to the On-line Manual.

Every category of help is connected to related keywords by hyperlink, and all the detailed explanations and information in connection with the keyword may be obtained.




***On-line Manual of midas FEA***

A summary of the help contents and an index of the main keywords are arranged systematically in the ***On-line Manual*** of midas FEA. The user should read it as a reference in the order presented in the summary. Alternatively, the information regarding the desired item may be directly obtained using the ***Search*** function of the keywords.