midas NFX

Understand different mesh types

This guide introduces different mesh types, how to choose between 1D, 2D, 3D elements, and how to assign element properties in midas NFX

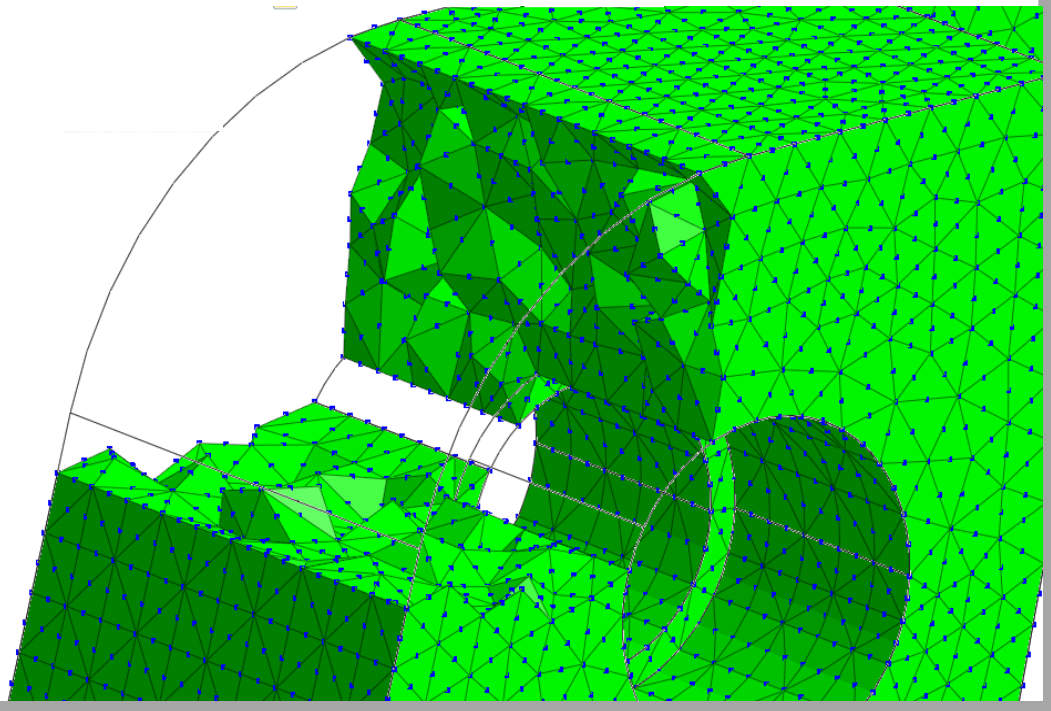


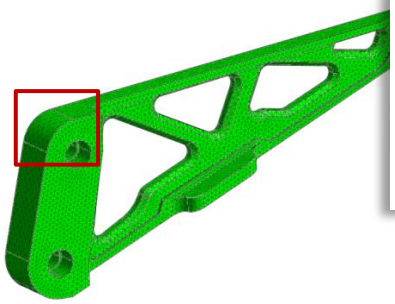
TABLE OF CONTENTS:

1 Reminder about meshes

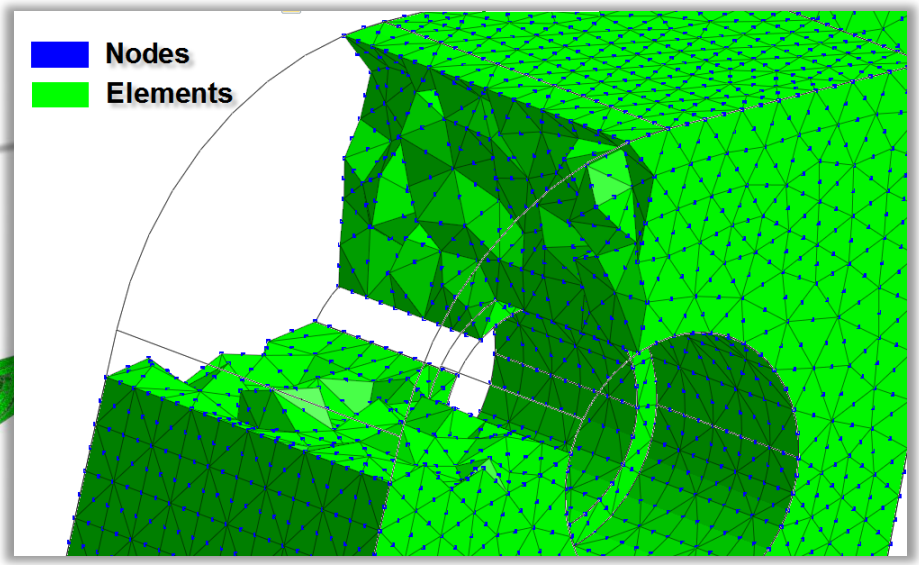
The Finite Element Method (FEM) is a numerical approximation method, which investigates the behavior of complex structures by breaking it down into smaller, simpler pieces.

These smaller pieces are called (finite) elements. The elements are connected to each other at nodes. These **elements and nodes are also called meshes**. The assembly of elements and nodes is called a finite element model.

CAD model

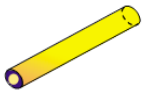
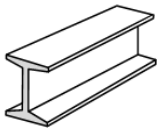
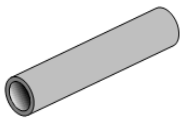
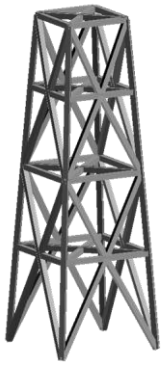
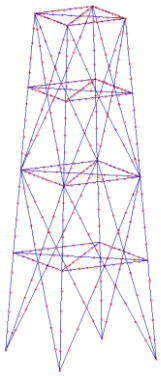
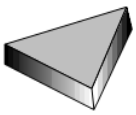
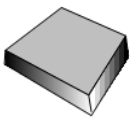
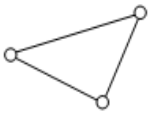
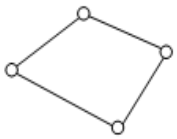
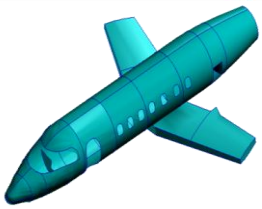
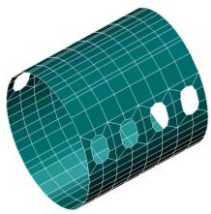
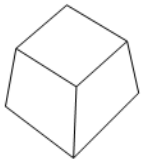
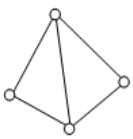
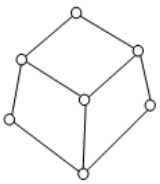
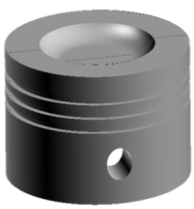
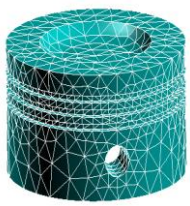


FEA model



2 Different mesh types

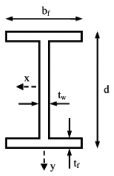
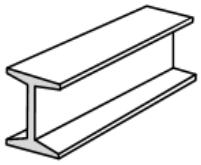
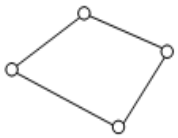
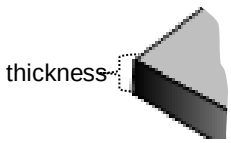
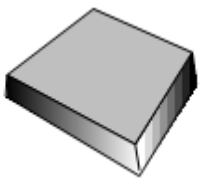
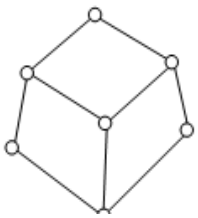
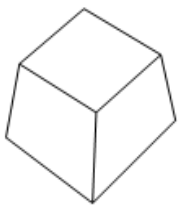
Mesh elements can be 1D, 2D or 3D elements in function of the model to simulate.

	Geometry	Model name	Finite element	Example
1D	  	Bar/Truss Beam Tube/Pipe	  	 
2D	 	Plate	 	 
3D	 	Solid	 	 

3 Element properties

FEA model is always a 3 dimensional model, so if 1D or 2D elements are used, 1 or 2 dimensions need to be defined in order to be analyzed.

These dimensions are defined by element properties.

Elements Dimensions	Property Dimensions	Total Dimensions
1D element 	Section property (2 constant dimensions) 	3 Dimensions 
2D element 	Thickness property (1 constant dimension) 	3 Dimensions 
3D element 	Solid property (0 dimension)	3 Dimensions 

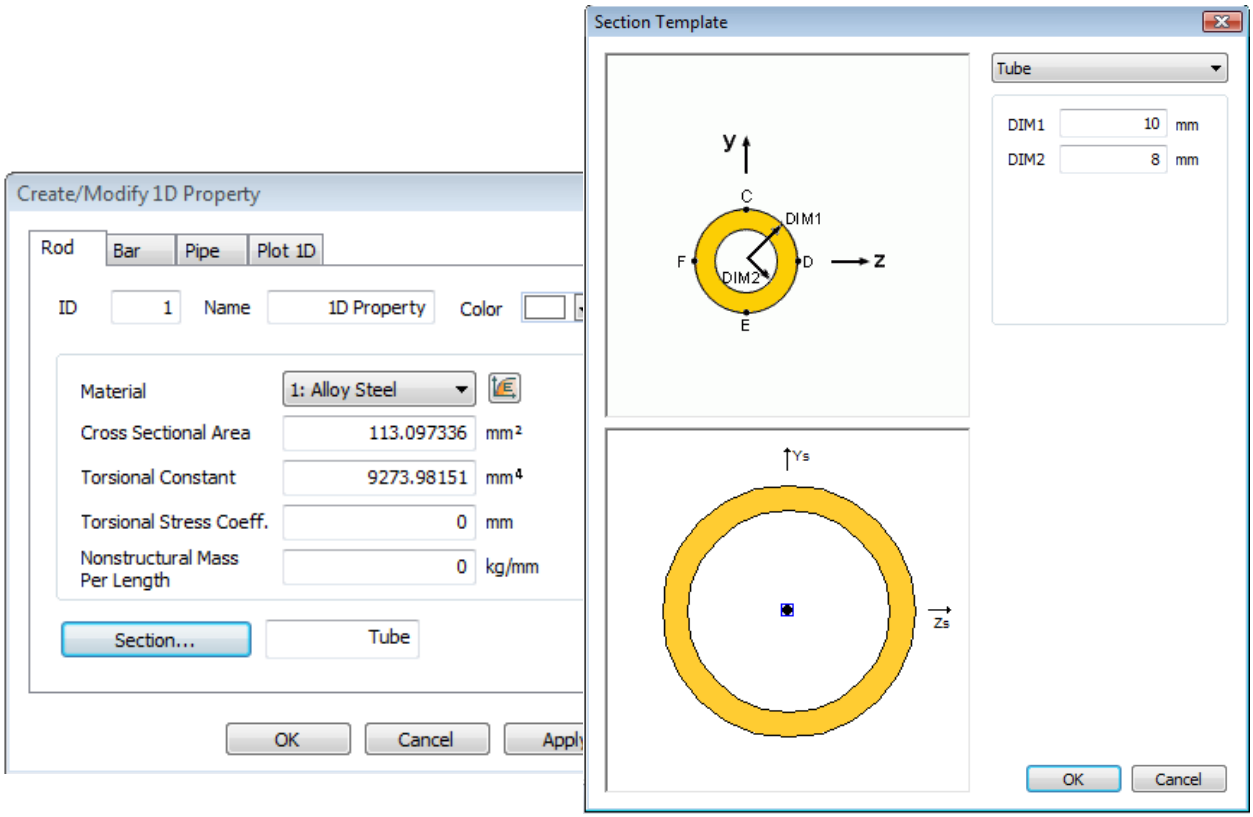
1D element properties

1D elements are used when length of the structure is 8~10 times greater than its width and thickness and when external loads are applied only to the joints.

There are 3 types of 1D elements:

- Rod element: only undertake compression
- Truss(bar) element: undertake tension and compression
- Beam element: undertake tension, compression, shear and moment

Section and material properties need to be assigned for elements.



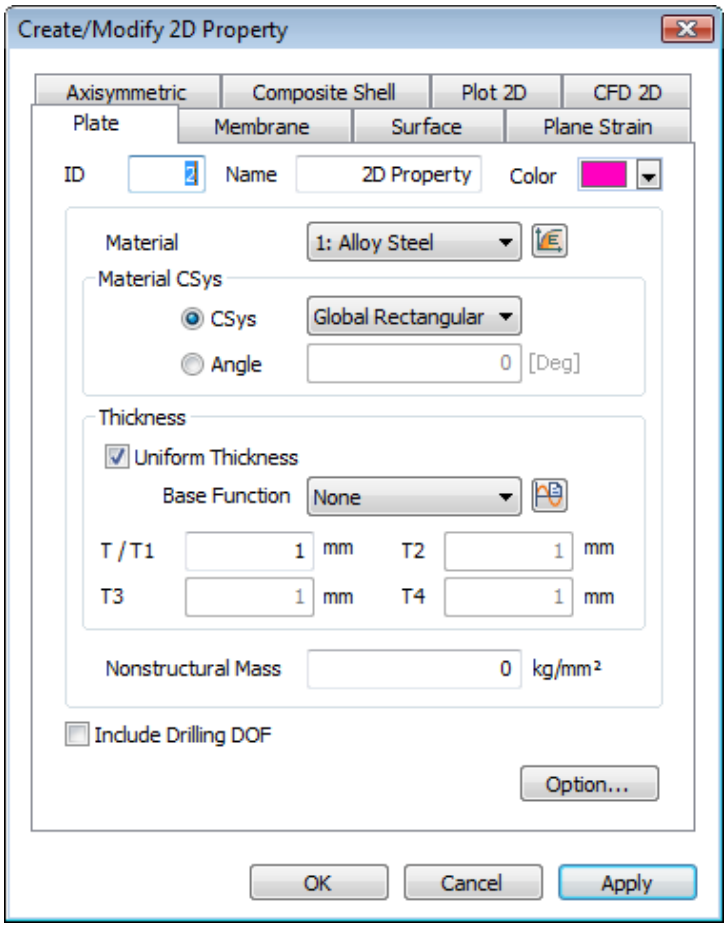
2D element properties

2D elements are used when length and width of the structure is significantly larger than it's thickness.

There are different types 2D elements:

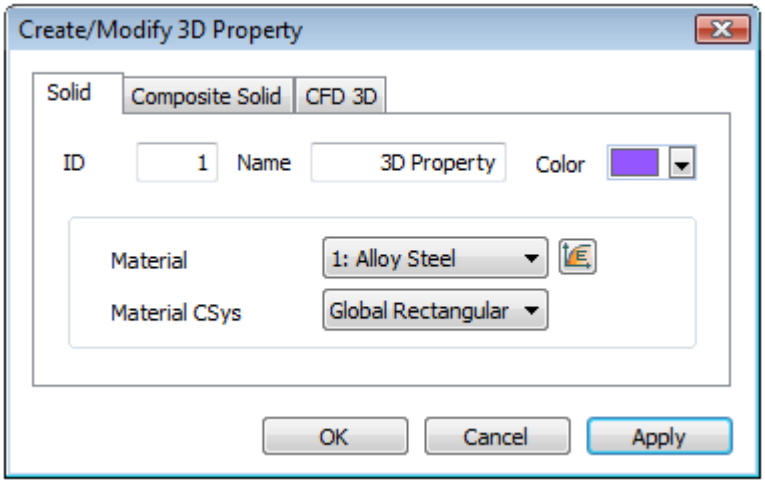
- Plate : general 2D element
- Membrane: no bending
- Surface: no thickness
- Plane strain: strain normal to the surface is 0
- Composite Shell: 2d plate elements with composite layers

Thickness and material properties need to be assigned for elements



3D element properties

3D element properties are only present to assign Materials and Material Coordinate Systems to 3D meshes.

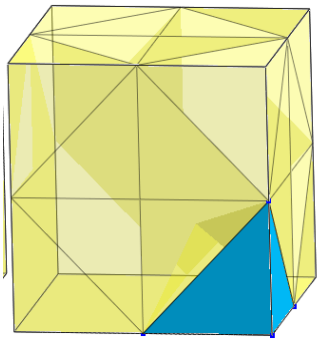


4 3D mesh types

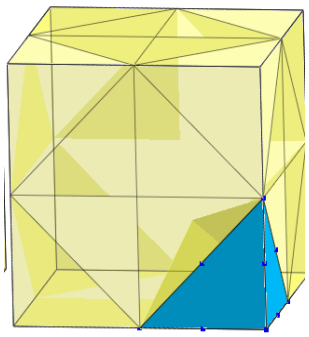
There are 2 main types of 3D elements, **tetrahedral elements** (4 faces, 4 nodes) and **hexahedral elements** (6 faces, 8 nodes).

2nd order elements, called also quadratic elements, add 1 middle node to each edge.

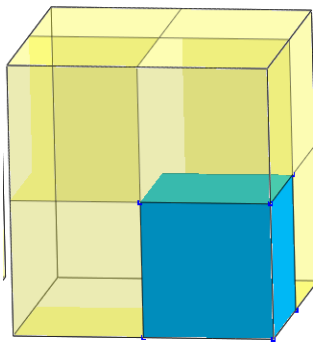
The calculation is more accurate for hexahedral elements than tetrahedral elements. And it's more accurate for 2nd order elements because the approximation function will be quadratic instead of linear.



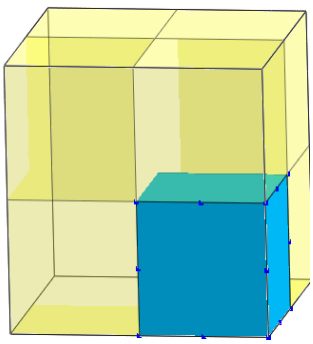
1st order tetra mesh (4 nodes)



2nd order tetra mesh (10 nodes)



1st order hexa mesh (8 nodes)



2nd order hexa mesh (20 nodes)

“Now you are ready to mesh your own FEA model !”

Please contact us if you have any question during your trial:

Email: Info@MidasUser.com

Telephone: +82-31-789-4040