Agenda

Who am I?
What you will learn
Femap capabilities
Demonstrations
Benefits of this topic
How to learn more
Meshing in FEMAP 0D

0D – Meshing

Mass Elements

1. Create Mass Property manually and attach to nodes

2. Use Tools, Mass Properties, Solid Properties
   - Select Solid
   - Answer “Yes” to “Ok to create a representative node and mass element”
   - Enter Mass Density
Meshing in FEMAP 1D

1D – Meshing
FEMAP “Line Elements”
Manual Creation

Pick two Nodes – Defines X Axis
Node or Vector for Cross-Section Alignment – Defines Y and Z Axes
Reference Point or Vector Offsets to define relation of Cross Section to selected Nodes
Meshing in FEMAP 1D – Mesh Between

Still around since the DOS version of FEMAP – Mesh, Between does exactly that, lays down elements between two user select points

Based on the Coordinate System selected, rectilinearly, cylindrically, or spherically
Reference Point is the most powerful and useful feature when meshing with 1-D elements in FEMAP.

Controls where automatic element offsets will be created to align the Reference Point with the element’s nodes, or with the geometric curve used to mesh on.
Meshing in FEMAP – 1D – On Geometry

Mesh, Mesh Control, Attributes On Curve

- Pre-Assign Property, Orientation, and Offsett, including using the Reference Point offset
This automatic handling of beam offsets and orientations saves you a lot of time in bookkeeping – NASTRAN expects the nodes to be at the Shear Center of the Beam, but the nodes are where the nodes are, FEMAP automatically sets up the offset from the nodes to the Shear Center on the CBEAM card, and the offset from the Shear Center to the Neutral Axis on the PBEAM card.
Meshing in FEMAP – 1D - Updates

Modify, Update Elements
Update –
• Orientation
• Offsets
• Reverse

Modify, Update Elements
Update –
• Orientation
• Offsets
• Reverse

Modify, Update Elements
Update –
• Orientation
• Offsets
• Reverse

Modify, Update Elements
Update –
• Orientation
• Offsets
• Reverse
API – Custom Updates

Since a Vector only points in one direction, all elements created at the same time are aligned with that Vector.

FEMAP ships with an API that can update beam orientations to align normal with a selected surface.
Meshing in FEMAP – 1D - Connections
Meshing in FEMAP – 1D - Connections

Mesh Closest Link finds the closest node for each node in the first selection.

If the first selection has more nodes that the second, you will get multiple elements at some of the nodes.

Mesh, Connect, Multiple collects the same setup information, and let’s the user pick the two nodes for connection.
Meshing in FEMAP – 1D – Unzip & Coincident Link

Unzip disconnects a Shell Mesh leaving coincident nodes.

In relation to 1-D Meshing, can also automatically create a 1-D connection between the original nodes and the new nodes.

Coincident Link uses FEMAP Coincident Node checking functionality and instead of merging the nodes, can automatically create 1-D connection.
Mesh, Edge Members

Meshes the Free Edges of a Shell Mesh with any 1-D Line Element

Select the Shell Elements to use

Pick nodes to define which free edges of the shells get the 1-D Line elements

Later on we’ll see how this same command can create shell elements on the free faces of a solid mesh
Meshing in FEMAP 2D – Mesh Between

Still around since the DOS version of FEMAP – Mesh, Between does exactly that, lays down 2-D elements between 3 or 4 user selected points. Based on the Coordinate System selected, rectilinearly, cylindrically, or spherically.
Meshing in FEMAP – 2D – Quadrilateral

Mapped Meshing Options

- When possible, FEMAP will lay down a “mapped” m x n mesh, the meshing algorithm will try to find four corners and mesh a parametrically re
Meshing in FEMAP – 2D – Mapped Meshing

When “Mapped Meshing” is turned on, FEMAP will also use hard-coded transition meshes between fully mapped areas.
Quad Edge Layers – attempts to lay down perfect layers of quads on the periphery of a quad mesh, when things get too twisted, it stops. The remainder of the mesh is then free-meshed.
Meshing in FEMAP – 2D – Options

Post-Mesh Cleanup – Looks for certain skewed and undesirable mesh patterns and replaces with better shaped elements – have not see a case where it’s not best to have this option on
"Min Elements Between Boundaries" forces the number of nodes entered between any cuts made by FEMAP’s boundary mesher.
Meshing in FEMAP – 2D - Options

“Cut Quads with Angle Deviation Above” – enter the maximum angle acceptable in a transition quad before FEMAP automatically splits it into two triangles. The higher the angle the less triangles in a mesh.
Three triangle meshers in FEMAP.

“Auto” – tells FEMAP to use the best one

3-D surfaces it will use the 3-D Tri
Planar Surfaces use Fast Tri

Subdivision is the oldest Tri-Mesher and is used if any of the others fail
Meshing in FEMAP – 2D – Triangle Meshing

Available in Quad Meshing, but more appropriate to triangle meshing – “Growth Factor” during sizing...

Very useful when optimizing total model size for large tetra meshes
Meshing in FEMAP – 3D - Tetrahedral

All Tetrahedral Meshing in FEMAP starts with a triangular surface mesh

- Triangles are fed into the INRIA tetrahedral mesher (used by most FEA vendors)
- Returned Tetrahedron exactly match the faces of the original triangles
Tetra Mesher handles internal triangles or quads, and creates a combined shell / solid mesh with slight embed for moment transfer.
Internal and branching faces and curves are accounted for within a solid

- Create a consistent mesh in volumes containing 3D, 2D, and 1D entities

Example:

Reinforced concrete model: steel beam elements with concrete tetra elements in a matched mesh
Hexahedral Meshing

Subdivide and Hex mesh
Mesh – Copy, Rotate, Reflect
Mesh – Extrude, Revolve, Sweep

Curves to Plates
Line Elements to Plate Elements
Plate Elements to Solid Elements
Problematic geometry

- Analysts inherit CAD models that often contain difficult geometry
  - Serviceable for CAD, but cause problems with FE meshing
  - Affects mesh and element quality and hence solution accuracy
- Typical problems include:
  - Sliver surfaces
  - Short edges
  - Narrow regions
  - Close points
  - Internal voids
- Difficult geometry has to be fixed before the model can be meshed
  - Manual geometry clean up requires much effort and time
Automatic Solid Geometry Preparation
Geometry and Meshing Considerations

Too much geometry detail
• CAD models often contain too much modeling detail for FEA
  • Small features that are not required for analysis
  • Must be removed before meshing

The solution:
• Femap’s automatic geometry preparation capability
  • Greatly reduces and in many cases completely eliminates the need for manual combination, suppression and splitting of geometry
Automatic Solid Geometry Preparation

- Automatic clean up allows:
  - Easier meshing of complex 3D models with problematic geometry
  - Easy removal of unnecessary detail
- Geometry preparation actions performed automatically include:
  - Combination of surfaces and curves
  - Suppression of small curves and surfaces
  - Surface splitting at strategic locations
  - User defined tolerance controls extent of geometry changes
Automatic Solid Geometry Preparation

- Prepare Geometry and Mesh sizing tolerances
  - Guide geometry feature removal / suppression and mesh sizing
- Internal voids identified and removed
- Access to further geometry preparation and mesh sizing options
Automatic Solid Geometry Preparation

- Full control over the extent of model simplification
- Identify a group of entities that should be ignored
- Entities that are loaded or constrained are ignored
- Set maximum sizes and angles based on geometry preparation tolerance
- Identify individual operations to be included in the cleanup process
Meshing in FEMAP

The Meshing Toolbox – Live Discussion/Demonstration