C O N T E N T S

MSC.Patran Reference Manual
Part 3: Finite Element Modeling

CHAPTER

1

Introduction to Finite Element Modeling

- General Definitions, 2
- How to Access Finite Element Modeling, 5
- Building a Finite Element Model for Analysis, 6
- Helpful Hints, 7
- Features in MSC.Patran for Creating the Finite Element Model, 8

2

The Create Action (Mesh)

- Introduction, 12
  - Element Topology, 13
  - Meshing Curves, 14
  - Meshing Surfaces with IsoMesh or Paver, 15
  - Meshing Solids, 17
  - Mesh Seeding, 19
  - Surface Mesh Control, 20
  - Remeshing and Reseeding, 21
- Mesh Seed and Mesh Forms, 29
  - Creating a Mesh Seed, 30
    - Uniform Mesh Seed, 30
    - One Way Bias Mesh Seed, 31
    - Two Way Bias Mesh Seed, 32
    - Curvature Based Mesh Seed, 33
    - Tabular Mesh Seed, 34
    - PCL Function Mesh Seed, 36
- Creating a Mesh, 38
  - IsoMesh Curve, 38
  - IsoMesh 2 Curves, 39
  - IsoMesh Surface, 40
    - Property Sets, 41
    - Create New Property, 42
    - Paver Parameters, 43
  - Solid, 44
    - IsoMesh, 44
    - TetMesh, 47
    - Node Coordinate Frames, 50
  - Mesh On Mesh, 51
    - Feature Select , 53
  - Sheet Body, 55
    - Feature Select, 57

- Advanced Surface Meshing, 58
  - Application Form, 58
  - 4. Final Mesh, 73
  - Final Mesh / Hard Geometry, 74
  - Display Small Entities, 76
  - Feature Properties, 77
- Mesh Control, 78
  - Auto Hard Points Form, 79

3
The Create Action (FEM Entities)

- Introduction, 84
- Creating Nodes, 84
  - Create Node Edit, 84
  - Create Node ArcCenter, 86
  - Extracting Nodes, 88
  - Interpolating Nodes, 95
  - Intersecting Two Entities to Create Nodes, 100
  - Creating Nodes by Offsetting a Specified Distance, 104
  - Piercing Curves Through Surfaces to Create Nodes, 107
  - Projecting Nodes Onto Surfaces or Faces, 109
- Creating Elements, 112
- Creating MPCs, 113
  - Create MPC Form (for all MPC Types Except Cyclic Symmetry and Sliding Surface), 117
    - Define Terms Form, 118
  - Create MPC Cyclic Symmetry Form, 119
  - Create MPC Sliding Surface Form, 120
- Creating Superelements, 122
  - Select Boundary Nodes, 123
- Creating DOF List, 124
  - Define Terms, 125
- Creating Connectors, 126
    - Spot Weld Properties Form, 128

4
The Transform Action

- Overview of Finite Element Modeling Transform Actions, 134
- Transforming Nodes, 135
  - Create Nodes by Translating Nodes, 135
  - Create Nodes by Rotating Nodes, 136
  - Create Nodes by Mirroring Nodes, 138
- Transforming Elements, 140
  - Create Elements by Translating Elements, 140
  - Create Elements by Rotating Elements, 141
  - Create Elements by Mirroring Elements, 142

5
The Sweep Action

- Introduction, 144
- Sweep Forms, 145
  - The Arc Method, 146
  - The Extrude Method, 147
  - The Glide Method, 148
    - Glide Control, 149
  - The Glide-Guide Method, 150
    - Glide-Guide Control, 152
  - The Normal Method, 153
  - The Radial Cylindrical Method, 154
  - The Radial Spherical Method, 155
  - The Spherical Theta Method, 156
  - The Vector Field Method, 158
  - The Loft Method, 160
    - FEM Data, 161
    - Mesh Control Data, 162

6
The Renumber Action

- Introduction, 166
- Renumber Forms, 167
  - Renumber Nodes, 168
  - Renumber Elements, 169
  - Renumber MPCs, 170

7
The Associate Action

- Introduction, 172
- Associate Forms, 173
  - The Point Method, 174
  - The Curve Method, 175
  - The Surface Method, 176
  - The Solid Method, 177
  - The Node Forms, 178

8
The Disassociate Action

- Introduction, 180
- Disassociate Forms, 181
  - Elements, 182
  - Node, 183

9
The Equivalence Action

- Introduction to Equivalencing, 186
- Equivalence Forms, 188

- Equivalence - All, 189
- Equivalence - Group, 190
- Equivalence - List, 191

10
The Optimize Action

- Introduction to Optimization, 194
- Optimizing Nodes and Elements, 196
- Selecting an Optimization Method, 197

11
The Verify Action

- Introduction to Verification, 200
- Verify Forms, 201
  - Verify - Element (Boundaries), 204
  - Verify - Element (Duplicates), 205
  - Verify - Element (Normals), 206
  - Verify - Element (Connectivity), 207
  - Verify - Element (Geometry Fit), 208
  - Verify - Element (Jacobian Ratio), 209
  - Verify - Element (Jacobian Zero), 210
  - Verify - Element (IDs), 211
  - Verify - Tria (All), 212
  - Verify - Tria (All) Spreadsheet, 213
  - Verify - Tria (Aspect), 214
  - Verify - Tria (Skew), 215
  - Verify - Quad (All), 216
  - Verify - Quad (All) Spreadsheet, 217
  - Verify - Quad (Aspect), 218
  - Verify - Quad (Warp), 219
  - Verify - Quad (Skew), 220
  - Verify - Quad (Taper), 221
  - Verify - Tet (All), 222
  - Verify - Tet (All) Spreadsheet, 223
  - Verify - Tet (Aspect), 224
  - Verify - Tet (Edge Angle), 225
  - Verify - Tet (Face Skew), 226
  - Verify - Tet (Collapse), 227
  - Verify - Wedge (All), 228
  - Verify - Wedge (All) Spreadsheet, 229
  - Verify - Wedge (Aspect), 230
  - Verify - Wedge (Edge Angle), 231
  - Verify - Wedge (Face Skew), 232
  - Verify - Wedge (Face Warp), 233
  - Verify - Wedge (Twist), 234
  - Verify - Wedge (Face Taper), 235
  - Verify - Hex (All), 236
  - Verify - Hex (All) Spreadsheet, 237
  - Verify - Hex (Aspect), 238
  - Verify - Hex (Edge Angle), 239
  - Verify - Hex (Face Skew), 240

- Verify - Hex (Face Warp), 241
- Verify - Hex (Twist), 242
- Verify - Hex (Face Taper), 243
- Verify - Node (IDs), 244
- Verify - Midnode (Normal Offset), 245
- Verify - Midnode (Tangent Offset), 246
- Superelement, 247

■ Theory, 248
- Skew, 248
- Aspect Ratio, 251
- Warp, 255
- Taper, 256
- Edge Angle, 257
- Collapse, 259
- Twist, 260

12
The Show Action

■ Show Forms, 264
- Show - Node Location, 265
- Show - Node Distance, 266
- Show - Element Attributes, 267
  - Write to Report, 268
- Show - Element Coordinate System, 269
- Show - Mesh Seed Attributes, 270
- Show - Mesh Control Attributes, 271
- Show - MPC, 272
  - Show - MPC Terms, 273
- Show Connectors, 274

13
The Modify Action

■ Introduction to Modification, 278

■ Modify Forms, 279
- Modifying Mesh, 280
  - Smoothing Parameters, 281
- Mesh Improvement Form, 283
  - General Parameters, 284
  - Process Control, 285
  - Collapse Ratio, 286
  - Jacobian Minimum, 287
- Modifying Mesh Seed, 288
- Sew Form, 289
- Modifying Elements, 291
  - Edit Method, 291
  - Reverse Method, 292
  - Separate Method, 293
  - Shell Orientation, 294
  - Display Control, 297
  - Fringe Attributes, 298
- Modifying Bars, 299

- Modifying Trias, 300
  - Splitting a Tria into Two Trias, 300
  - Splitting a Tria into Three Trias, Four Trias, or Three Quads, 301
  - Splitting a Tria into a Tria and a Quad, 302
  - Splitting Tet Elements, 303
- Modifying Quads, 304
  - Splitting a Quad into Two Quads, 304
  - Splitting a Quad into Three Quads, 305
  - Splitting a Quad into Four Quads or Four Trias or NxM Quads, 306
  - Splitting a Quad into Two Trias, 307
  - Splitting a Quad into Three Trias, 308
- Modifying Nodes, 309
  - Move Method, 309
  - Offset Method, 310
  - Edit Method, 311
  - Project Method, 312
- Modifying MPCs, 313
  - Modify Terms, 314
- Modifying Spot Weld Connectors, 315

14
The Delete Action

- Delete Action, 318
- Delete Forms, 319
  - Delete - Any, 320
  - Delete - Mesh Seed, 321
  - Delete - Mesh (Surface), 322
  - Delete - Mesh (Curve), 323
  - Delete - Mesh (Solid), 324
  - Delete - Mesh Control, 325
  - Delete - Node, 326
  - Delete - Element, 327
  - Delete - MPC, 328
  - Delete - Connector, 329
  - Delete - Superelement, 330
  - Delete - DOF List, 331

15
The MSC.Patran Element Library

- Introduction, 334
- Beam Element Topology, 336
- Tria Element Topology, 338
- Quad Element Topology, 345
- Tetrahedral Element Topology, 351
- Wedge Element Topology, 364
- Hex Element Topology, 381
- MSC.Patran's Element Library, 394

INDEX

- MSC.Patran Reference Manual, 409

Part 3: Finite Element Modeling

CHAPTER

1

Introduction to Finite Element Modeling

- General Definitions
- How to Access Finite Element Modeling
- Building a Finite Element Model for Analysis
- Helpful Hints
- Features in MSC.Patran for Creating the Finite Element Model

1.1 General Definitions

analysis coordinate frame	A local coordinate system associated to a node and used for defining constraints and calculating results at that node.
attributes	ID, topology, parent geometry, number of nodes, applied loads and bcs, material, results.
connectivity	The order of nodes in which the element is connected. Improper connectivity can cause improperly aligned normals and negative volume solid elements.
constraint	A constraint in the solution domain of the model.
cyclic symmetry	A model that has identical features repeated about an axis. Some analysis codes such as MSC.Nastran explicitly allow the identification of such features so that only one is modeled.
degree-of-freedom	DOF, the variable being solved for in an analysis, usually a displacement or rotation for structural and temperature for thermal at a point.
dependent DOF	In an MPC, the degree-of-freedom that is condensed out of the analysis before solving the system of equations.
equivalencing	Combining nodes which are coincident (within a distance of tolerance) with one another.
explicit	An MPC that is not interpreted by the analysis code but used directly as an equation in the solution.
finite element	1. A general technique for constructing approximate solutions to boundary value problems and which is particularly suited to the digital computer. 2. The MSC.Patran database entities point element, bar, tria, quad, tet, wedge and hex.
finite element model	A geometry model that has been descritized into finite elements, material properties, loads and boundary conditions, and environment definitions which represent the problem to be solved.
free edges	Element edges shared by only one element.
free faces	Element faces shared by only one element.
implicit	An MPC that is first interpreted into one or more explicit MPCs prior to solution.
independent DOF	In an MPC, the degree-of-freedom that remains during the solution phase.
IsoMesh	Mapped meshing capability on curves, three- and four-sided biparametric surfaces and triparametric solids available from the Create, Mesh panel form.

Jacobian Ratio	The ratio of the maximum determinant of the Jacobian to the minimum determinant of the Jacobian is calculated for each element in the current group in the active viewport. This element shape test can be used to identify elements with interior corner angles far from 90 degrees or high order elements with misplaced midside nodes. A ratio close or equal to 1.0 is desired.
Jacobian Zero	The determinant of the Jacobian (J) is calculated at all integration points for each element in the current group in the active viewport. The minimum value for each element is determined. This element shape test can be used to identify incorrectly shaped elements. A well-formed element will have J positive at each Gauss point and not greatly different from the value of J at other Gauss points. J approaches zero as an element vertex angle approaches 180 degrees.
library	Definition of all element topologies.
MPC	Multi-Point Constraint. Used to apply more sophisticated constraints on the FEM model such as sliding boundary conditions.
non-uniform seed	Uneven placement of node locations along a curve used to control node creation during meshing.
normals	Direction perpendicular to the surface of an element. Positive direction determined by the cross-product of the local parametric directions in the surface. The normal is used to determine proper orientation of directional loads.
optimization	Renumbering nodes or elements to reduce the time of the analysis. Applies only to wavefront or bandwidth solvers.
parameters	Controls for mesh smoothing algorithm. Determines how fast and how smooth the resulting mesh is produced.
paths	The path created by the interconnection of regular shaped geometry by keeping one or two constant parametric values.
Paver	General meshing of n-sided surfaces with any number of holes accessed from the Create / Mesh / Surface panel form.
reference coordinate frame	A local coordinate frame associated to a node and used to output the location of the node in the Show, Node, Attribute panel. Also used in node editing to define the location of a node.
renumber	Change the IDs without changing attributes or associations.
seeding	Controlling the mesh density by defining the number of element edges along a geometry curve prior to meshing.
shape	The basic shape of a finite element (i.e., tria or hex).
sliding surface	Two surfaces which are in contact and are allowed to move tangentially to one another.
sub MPC	A convenient way to group related implicit MPCs under one MPC description.
term	A term in an MPC equation which references a node ID, a degree-of-freedom and a coefficient (real value).

Tetmesh General meshing of n-faced solids accessed from the Create / Mesh / Solid panel form.

topology The shape, node, edge, and face numbering which is invariant for a finite element.

transitions The result of meshing geometry with two opposing edges which have different mesh seeds. Produces an irregular mesh.

types For an implicit MPC, the method used to interpret for analysis.

uniform seed Even placement of nodes along a curve.

verification Check the model for validity and correctness.

1.2 How to Access Finite Element Modeling

The Finite Elements Application. All of MSC.Patran's finite element modeling capabilities are available by selecting the Finite Element button on the main form. Finite Element (FE) Meshing, Node and Element Editing, Nodal Equivalencing, ID Optimization, Model Verification, FE Show, Modify and Delete, and ID Renumber, are all accessible from the Finite Elements form.

At the top of the form are a set of pull-down menus named Action and Object, followed by either Type, Method or Test. These menus are hierarchical. For example, to verify a particular finite element, the Verify action must be selected first. Once the type of Action, Object and Method has been selected, MSC.Patran will store the setting. When the user returns to the Finite Elements form, the previously defined Action, Object and Method will be displayed. Therefore, MSC.Patran will minimize the number steps if the same series of operations are performed.

The Action menu is organized so the following menu items are listed in the same order as a typical modeling session.

1. Create
2. Transform
3. Sweep
4. Renumber
5. Associate
6. Equivalence
7. Optimize
8. Verify
9. Show
10. Modify
11. Delete

1.3 Building a Finite Element Model for Analysis

MSC.Patran provides numerous ways to create a finite element model. Before proceeding, it is important to determine the analysis requirements of the model. These requirements determine how to build the model in MSC.Patran. Consider the following:

Table 1-1 Considerations in Preparing for Finite Element Analysis

Desired Response Parameters	Displacements, Stresses, Buckling, Combinations, Dynamic, Temperature, Magnetic Flux, Acoustical, Time Dependent, etc.
Scope of Model	Component or system (Engine mount vs. Whole Aircraft).
Accuracy	First "rough" pass or within a certain percent.
Simplifying Assumptions	Beam, shell, symmetry, linear, constant, etc.
Available Data	Geometry, Loads, Material model, Constraints, Physical Properties, etc.
Available Computational Resources	CPU performance, available memory, available disk space, etc.
Desired Analysis Type	Linear static, nonlinear, transient deformations, etc.
Schedule	How much time do you have to complete the analysis?
Expertise	Have you performed this type of analysis before?
Integration	CAD geometry, coupled analysis, test data, etc.

Table 1-1 lists a portion of what a Finite Element Analyst must consider before building a model.The listed items above will affect how the FEM model will be created. The following two references will provide additional information on designing a finite element model.

- NAFEMS. *A Finite Element Primer*. Dept. of Trade and Industry, National Engineering Laboratory, Glasgow,UK,1986.
- Schaeffer, Harry G, *MSC/NASTRAN Primer*. Schaeffer Analysis Inc., 1979.

In addition, courses are offered at MSC.Software Corporation's MSC Institute, and at most major universities which explore the fundamentals of the Finite Element Method.

1.4 Helpful Hints

If you are ready to proceed in MSC.Patran but are unsure how to begin, start by making a simple model. The model should contain only a few finite elements, some unit loads and simple physical properties. Run a linear static or modal analysis. By reducing the amount of model data, it makes it much easier to interpret the results and determine if you are on the right track.

Apply as many simplifying assumptions as possible. For example, run a 1D problem before a 2 D, and a 2D before a 3D. For structural analysis, many times the problem can be reduced to a single beam which can then be compared to a hand calculation.

Then apply what you learned from earlier models to more refined models. Determine if you are converging on an answer. The results will be invaluable for providing insight into the problem, and comparing and verifying the final results.

Determine if the results you produce make sense. For example, does the applied unit load equal to the reaction load? Or if you scale the loads, do the results scale?

Try to bracket the result by applying extreme loads, properties, etc. Extreme loads may uncover flaws in the model.

1.5 Features in MSC.Patran for Creating the Finite Element Model

Table 1-2 lists the four methods available in MSC.Patran to create finite elements.

Table 1-2 Methods for Creating Finite Elements in MSC.Patran

IsoMesh	Traditional mapped mesh on regularly shaped geometry. Supports all elements in MSC.Patran.
Paver	Surface mesher. Can mesh 3D surfaces with an arbitrary number of edges and with any number of holes. Generates only area, or 2D elements.
Editing	Creates individual elements from previously defined nodes. Supports the entire MSC.Patran element library. Automatically generates midedge, midface and midbody nodes.
TetMesh	Arbitrary solid mesher generates tetrahedral elements within MSC.Patran solids defined by an arbitrary number of faces or volumes formed by collection of triangular element shells. This method is based on MSC plastering technology.

Isomesh. The IsoMesh method is the most versatile for creating a finite element mesh. It is accessed by selecting:

Action: Create

Object: Mesh

IsoMesh will mesh any untrimmed, three- or four-sided set of biparametric (green) surfaces with quadrilateral or triangular elements; or any triparametric (blue) solids with hedahedral, wedge or tetrahedral elements.

Mesh density is controlled by the “Global Edge Length” parameter on the mesh form. Greater control can be applied by specifying a mesh seed which can be accessed by selecting:

Action: Create

Object: Mesh Seed

Mesh seeds are applied to curves or edges of surfaces or solids. There are options to specify a uniform or nonuniform mesh seed along the curve or edge.

Paver. Paver is used for any trimmed (red) surface with any number of holes. Paver is accessed in the same way as IsoMesh except the selected Object must be Surface. Mesh densities can be defined in the same way as IsoMesh. The mesh seed methods are fully integrated and may be used interchangeably for IsoMesh and Paver. The resulting mesh will always match at common geometric boundaries.

TetMesh. TetMesh is used for any solid, and is especially useful for unparametrized or b-rep (white) solids. TetMesh is accessed the same way as IsoMesh, except the selected Object must be Solid. Mesh densities can be defined in the same way as IsoMesh. The mesh seed methods are fully integrated and may be used interchangeably for IsoMesh and TetMesh. The resulting mesh will always match at common geometric boundaries.

MPC Create. Multi-point constraints (MPCs) provide additional modeling capabilities and include a large library of MPC types which are supported by various analysis codes. Perfectly rigid beams, slide lines, cyclic symmetry and element transitioning are a few of the supported MPC types available in MSC.Patran.

Transform. Translate, rotate, or mirror nodes and elements.

Sweep. Create a solid mesh by either extruding or arcing shell elements or the face of solid elements.

Renumber. The Finite Element application's Renumber option is provided to allow direct control of node and element numbering. Grouping of nodes and elements by a number range is possible through Renumber.

Associate. Create database associations between finite elements (and their nodes) and the underlying coincident geometry. This is useful when geometry and finite element models are imported from an outside source and, therefore, no associations are present.

Equivalencing. Meshing creates coincident nodes at boundaries of adjacent curves, surfaces, and/or solids. Equivalencing is an automated means to eliminate duplicate nodes.

Optimize. To use your computer effectively, it is important to number either the nodes or the elements in the proper manner. This allows you to take advantage of the computer's CPU and disk space for the analysis. Consult your analysis code's documentation to find out how the model should be optimized before performing MSC.Patran's Analysis Optimization.

Verification. Sometimes it is difficult to determine if the model is valid, such as, are the elements connected together properly? are they inverted or reversed? etc. This is true--even for models which contain just a few finite elements. A number of options are available in MSC.Patran for verifying a Finite Element model. Large models can be checked quickly for invalid elements, poorly shaped elements and proper element and node numbering. Quad element verification includes automatic replacement of poorly shaped quads with improved elements.

Show. The Finite Element application's Show action can provide detailed information on your model's nodes, elements, and MPCs.

Modify. Modifying node, element, and MPC attributes, such as element connectivity, is possible by selecting the Modify action. Element reversal is also available under the Modify action menu.

Delete. Deleting nodes, elements, mesh seeds, meshes and MPCs are available under the Finite Element application's Delete menu. You can also delete associated stored groups that are empty when deleting entities that are contained in the group.

CHAPTER

2

The Create Action (Mesh)

- Introduction
- Mesh Seed and Mesh Forms
- Creating a Mesh
- Mesh Control

2.1 Introduction

Mesh creation is the process of creating finite elements from curves, surfaces, or solids. MSC.Patran provides the following automated meshers: IsoMesh, Paver, and TetMesh.

IsoMesh operates on parametric curves, biparametric (green) surfaces, and triparametric (blue) solids. It can produce any element topology in the MSC.Patran finite element library.

Paver can be used on any type of surface, including n-sided trimmed (red) surfaces. Paver produces either quad or tria elements.

IsoMesh, Paver, and TetMesh provide flexible mesh transitioning through user-specified mesh seeds. They also ensure that newly meshed regions will match any existing mesh on adjoining congruent regions.

TetMesh generates a mesh of tetrahedral elements for any triparametric (blue) solid or B-rep (white) solid.

Element Topology

MSC.Patran users can choose from an extensive library of finite element types and topologies. The finite element names are denoted by a shape name and its number of associated nodes, such as Bar2, Quad4, Hex20. See **The MSC.Patran Element Library** (Ch. 15) for a complete list.

MSC.Patran supports seven different element shapes, as follows:

- **point**
- **bar**
- **tria**
- **quad**
- **tet**
- **wedge**
- **hex**

For defining a specific element, first choose analysis under the preference menu, and select the type of analysis code. Then select Finite Elements on the main menu, and when the Finite Elements form appears, define the element type and topology.

When building a MSC.Patran model for an external analysis code, it is highly recommended that you review the Application Preference Guide to determine valid element topologies for the analysis code before meshing.

Meshing Curves

Meshes composed of one-dimensional bar elements are based on the IsoMesh method and may be applied to curves, the edges of surfaces, or the edges of solids. For more information on IsoMesh, see **Meshing Surfaces with IsoMesh or Paver** (p. 15).

Bar or beam element orientations defined by the bar's XY plane, are specified through the assignment of an element property. For more information on defining bar orientations, see **Element Properties Application** (Ch. 3) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments*.

IsoMesh 2 Curves. This method will create an IsoMesh between two curve lists. The mesh will be placed at the location defined by ruling between the two input curves. The number of elements will be determined by global edge length or a specified number across and along. For more information on IsoMesh, see **Meshing Surfaces with IsoMesh or Paver** (p. 15).

Meshing Surfaces with IsoMesh or Paver

MSC.Patran can mesh a group of congruent surfaces (i.e., adjoining surfaces having shared edges and corner points). Both surfaces and faces of solids can be meshed. MSC.Patran provides a choice of using either the IsoMesh method or the Paver method depending on the type of surface to be meshed.

IsoMesh is used for parametrized (green) surfaces with *only three or four sides*.

Important:	Green surfaces may be constructed using chained curves with slope discontinuities and thus may appear to have more than four sides. During meshing, a node will be placed on any slope discontinuity whose angle exceeds the “Node/Edge Snap Angle.” See **Preferences>Finite Element** (p. 366) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Paver can mesh trimmed or untrimmed (red) surfaces with *more than four sides*, as well as parametric (green) surfaces.

IsoMesh. IsoMesh will create equally-spaced nodes along each edge in real space--even for nonuniformly parametrized surfaces. IsoMesh computes the number of elements and node spacing for every selected geometric edge before any individual region is actually meshed. This is done to ensure that the new mesh will match any existing meshes on neighboring regions.

IsoMesh requires the surfaces to be parametrized (green), and to have either *three or four sides*. Surfaces which have more than four sides must first be decomposed into smaller three- or four-sided surfaces. Trimmed (red) surfaces must also be decomposed into three- or four-sided surfaces before meshing with IsoMesh. For complex n-sided surfaces, the Paver is recommended.

For more information on decomposing surfaces, see **Building a Congruent Model** (p. 31) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*.

Mesh Paths. After selecting the surfaces to be meshed, IsoMesh divides the surfaces’ edges into *groups of topologically parallel edges* called *Mesh Paths*. Mesh Paths are used by IsoMesh to calculate the number of elements per edge based on either adjoining meshed regions, mesh seeded edges, or the global element edge length.

If a *mesh seed* is defined for one of the edges in the path, or there is an *adjoining meshed region* on one of the mesh path’s edges, IsoMesh will ignore the *global element edge length* for all edges in the path. IsoMesh will apply the same number of elements, along with the node spacing, from the adjoining meshed region or the mesh seeded edge to the remaining edges in the path.

IsoMesh will use the *global element edge length* for a mesh path if there are no neighboring meshed regions or mesh seeded edges within the path. IsoMesh will calculate the number of elements per edge by taking the *longest edge* in the mesh path and dividing by the global edge length, and rounding to the nearest integer value.

Figure 2-1 shows two adjoining surfaces with mesh paths A, B, and C defined by IsoMesh. Edge “1” is a member of mesh path A and has a mesh seed of five elements. Edge “2” is a member of mesh path B and has a mesh seed of eight elements. As shown in **Figure 2-2**, IsoMesh created five elements for the remaining edges in mesh path A, and eight elements for the remaining edge in mesh path B. Since there are no mesh seeds or adjoining meshes for mesh path C, IsoMesh uses the global element edge length to calculate the number of elements for each edge.

Paver. Paver is best suited for trimmed (red) surfaces, including complex surfaces with more than four sides, such as surfaces with holes or cutouts. See **Figure 2-7**.

Paver is also good for surfaces requiring “steep” mesh transitions, such as going from four to 20 elements across a surface. Similar to IsoMesh, the paver calculates the node locations in real space, but it does not require the surfaces to be parametrized.

Important:	For an all quadrilateral element mesh, the Paver requires the total number of elements around the perimeter of each surface to be an even number. It will automatically adjust the number of elements on a free edge to ensure this condition is met.

Meshing Solids

MSC.Patran meshes solids with the IsoMesh or TetMesh.

IsoMesh can mesh any group of congruent triparametric (blue) solids (i.e., adjoining solids having shared edges and corner points). Triparametric solids with the topological shape of a brick or a wedge can be meshed with either hex or wedge elements. Any other form of triparametric solid can only be meshed with tet elements. Solids that have more than six faces must first be modified and decomposed before meshing.

TetMesh can be used to mesh all (blue or white) solids in MSC.Patran.

Mesh Paths. Since IsoMesh is used to mesh solids, similar to meshing surfaces, Mesh Paths are used to determine the number of elements per solid edge. For more detailed information on Mesh Paths, see **Meshing Surfaces with IsoMesh or Paver** (p. 15).

If there is a preexisting mesh adjoining one of the edges or a defined mesh seed on one of the edges in a mesh path, MSC.Patran will apply the same number of elements to the remaining edges in the path. If there are no adjoining meshes or mesh seeds defined within a path, the global element edge length will be used to determine the number of elements.

Figure 2-3 shows two adjoining congruent solids with mesh Paths A, B, C, and D defined. Edge "1" of path A has a mesh seed of five elements. Edge "2" of path B has a mesh seed of fourteen elements. And Edge "3" of path C has a nonuniform mesh seed of six elements. See **Mesh Seeding** (p. 19) for more information.

Figure 2-4 shows the solid mesh. Since Mesh Path A has a seed of five elements, all edges in the path are also meshed with five elements. The same applies for Mesh Paths B and C, where the seeded edge in each path determines the number of elements and node spacing. Since Mesh Path D did not have a mesh seed, or a preexisting adjoining mesh, the global element edge length was used to define the number of elements.

TetMesh. TetMesh will attempt to mesh any solid with very little input from the user as to what size of elements should be created. Generally, this is not what is needed for an actual engineering analysis. The following tips will assist the user both in terms of getting a good quality mesh suitable for the analysis phase and also tend to improve the success of TetMesh. If TetMesh fails to complete the mesh and the user has only specified a Global Length on the form, success might still be obtained by following some of the suggestions below.

Try to mesh the surfaces of a solid with the Paver using tria elements. If the Paver cannot mesh the solid faces, it is unlikely that TetMesh will be able to mesh the solid. By paving the solid faces first, much better control of the final mesh can be obtained. The mesh can be refined locally as needed. The surface meshing may also expose any problems with the geometry that make it difficult or impossible to mesh. Then these problems can be corrected before undertaking the time and expense to attempt to TetMesh the solid.

If higher order elements are required from a surface mesh of triangular elements, the triangular elements must also be of the corresponding order so that the mid edge nodes would be snapped properly.

Tria meshes on the solid faces can be left on the faces and stored in the database. This allows them to be used in the future as controls for the tet mesh in the solid at a later time.

After the tria mesh is created on the solid faces, it should be inspected for poor quality tria elements. These poor quality elements typically occur because Paver meshed a small feature in the geometry that was left over from the construction of the geometry, but is not important to the analysis. If these features are removed prior to meshing or if the tria mesh is cleaned up prior to tet meshing, better success rates and better tet meshes will usually follow. Look for high aspect ratios in the tria elements and look for tria elements with very small area.

The following paragraph applies only to the State Machine Algorithm.

Once the solid faces have a tria mesh, TetMesh will match the tet element faces to the existing tria elements. Just select the solid as input to TetMesh. This is not the same as selecting the tria shell as input. By selecting the solid, the resulting tet mesh will be associated with the solid and the element mid-edge nodes on the boundary will follow the curvature of the geometry. Note that the tria mesh on the solid faces do not need to be higher order elements in order for a higher order tet mesh to snap its mid-edge nodes to the geometry.

Mesh Seeding

Mesh Transitions. A mesh transition occurs when the number of elements differs across two opposing edges of a surface or solid face. Mesh transitions are created either by *mesh seeding* the two opposing edges with a different number of elements, or by *existing meshes* on opposite sides of the surface or solid face, whose number of elements differ.

If IsoMesh is used for the transition mesh, MSC.Patran uses smoothing parameters to create the mesh. For most transition meshes, it is unnecessary to redefine the parameter values. See **IsoMesh Parameters Subordinate Form** (p. 45).

Seeding Surface Transitions. MSC.Patran can mesh a set of surfaces for any combination of mesh seeds. *A mesh transition can occur in both directions of a surface.*

Seeding Solid Transitions. Transition meshes for solids *can only occur in two of three directions* of the solids. That is, the transition can be defined on one side of a set of solids, and carried through the solids' third direction. If a transition is required in all three directions, the user must break one of the solids into two, and perform the transition in two steps, one in each sub-solid. If a set of solids are seeded so that a transition will take place in all three directions, MSC.Patran will issue an error and not mesh the given set of solids.

If more than one mesh seed is defined within a single mesh path (a mesh path is a group of topologically parallel edges for a given set of solids), it *must belong to the same solid face*. Otherwise, MSC.Patran will issue an error and not mesh the specified set of solids (see **Figure 2-5** and **Figure 2-6**). If this occurs, additional mesh seeds will be required in the mesh path to further define the transition. For more information on mesh paths, see **Mesh Solid** (p. 44).

Avoiding Triangular Elements. MSC.Patran will try to avoid inserting triangle elements in a quadrilateral surface mesh, or wedge elements in a hexagonal solid mesh.

However, if the total number of elements around the perimeter of a surface, or a solid face is an odd number, the IsoMesh method will *produce one triangular or one row of wedge element* per surface or solid. Remember IsoMesh is the default meshing method for solids, as well as for curves.

If the total number of elements around the surface's or solid's perimeter is even, IsoMesh will mesh the surface or solid with Quad or Hex elements only. If the surface or solid is triangular or wedge shaped, and the mesh pattern chosen on the **IsoMesh Parameters Subordinate Form** (p. 45) form is the triangular pattern, triangle or wedge elements will be created regardless of the number of elements.

Figure 2-8 through **Figure 2-13** show examples of avoiding triangular elements with IsoMesh.

When Quad elements are the desired element type, MSC.Patran's *Paver requires* the number of elements around the perimeter of the surface to be *even*. If the number is odd, an error will be issued and Paver will ask the user if he wishes to use tri elements for this surface. If Quad elements are desired, the user must readjust the mesh seeds to an even number before meshing the surface again.

Surface Mesh Control

Users can specify surface mesh control on selected surfaces to be used when meshing using any of the auto meshers. This option allows users to create meshes with transition without having to do so one surface at a time. This option is particularly useful when used with the solid tet mesher to create mesh densities that are different on the edge and on the solid surface.

Remeshing and Reseeding

An existing mesh or mesh seed does not need to be deleted before remeshing or reseeding. MSC.Patran will ask for permission to delete the existing mesh or mesh seed before creating a new one.

However, mesh seeds cannot be applied to edges with an existing mesh, unless the mesh seed will exactly match the number of elements and node spacing of the existing mesh. Users must first delete the existing mesh, before applying a new mesh seed to the edge.

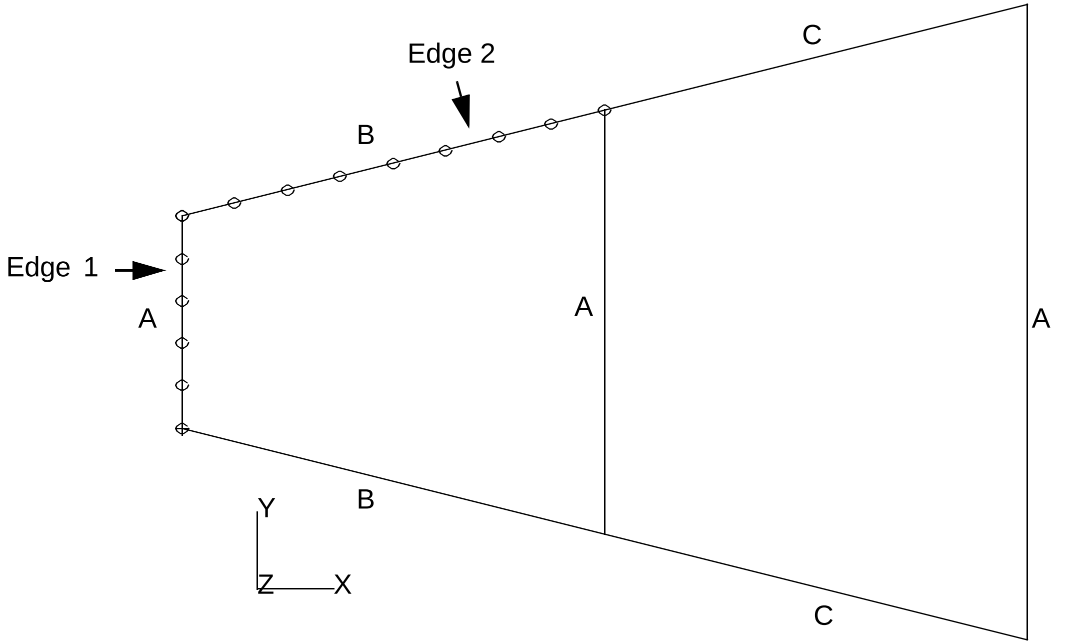


Figure 2-1 IsoMesh Mesh Paths A, B, C

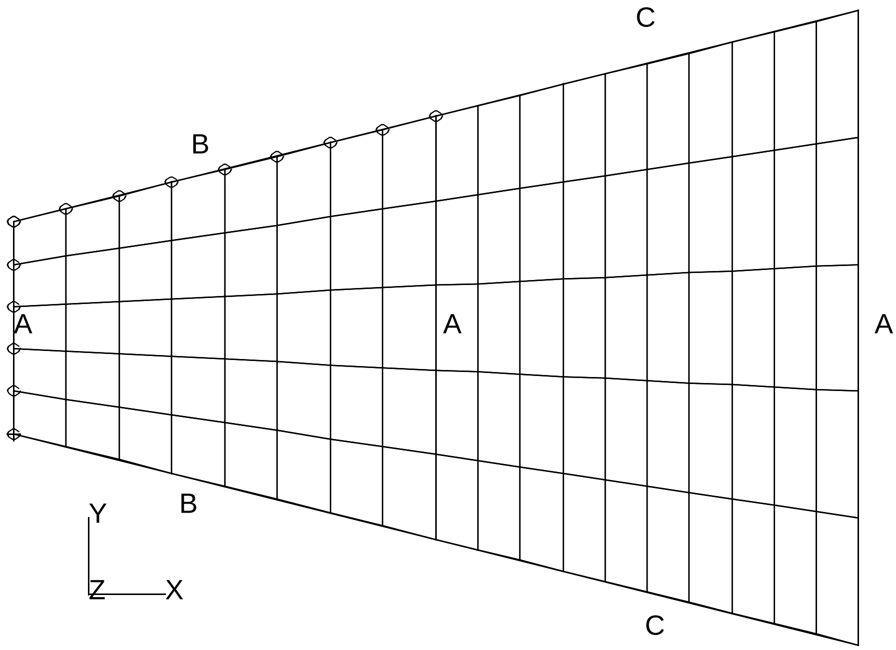


Figure 2-2 Meshed Surfaces Using IsoMesh

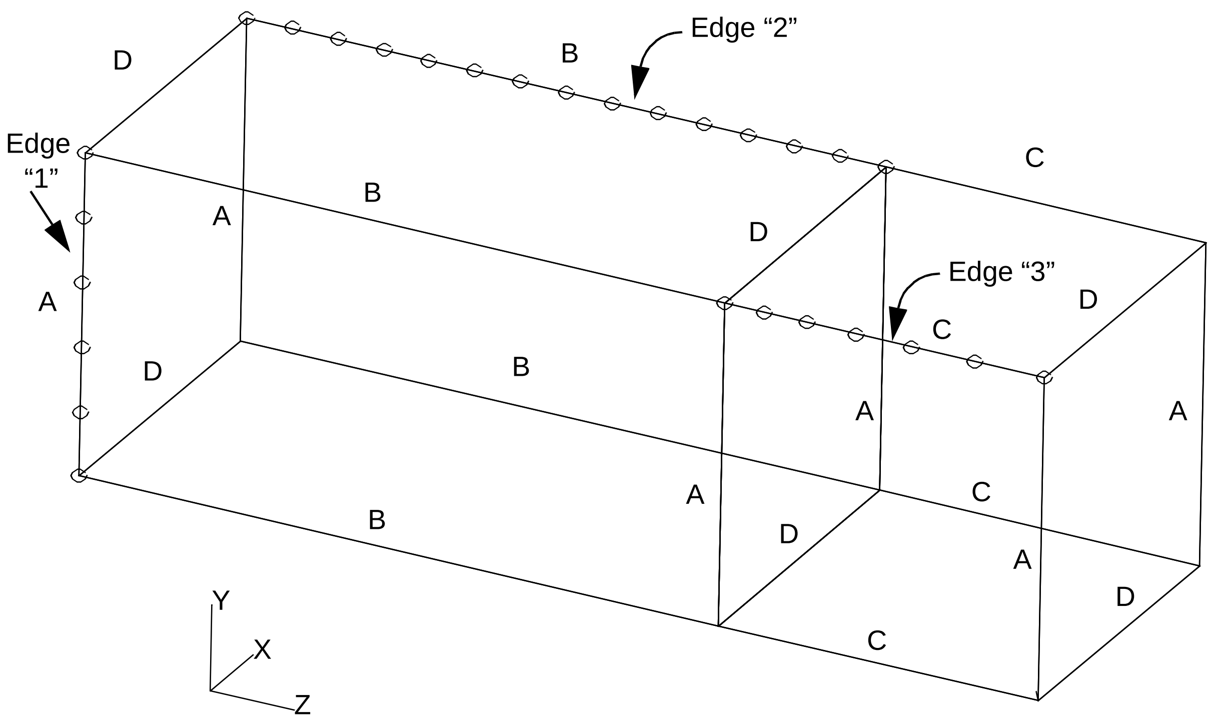


Figure 2-3 Mesh Seeding for Two Solids

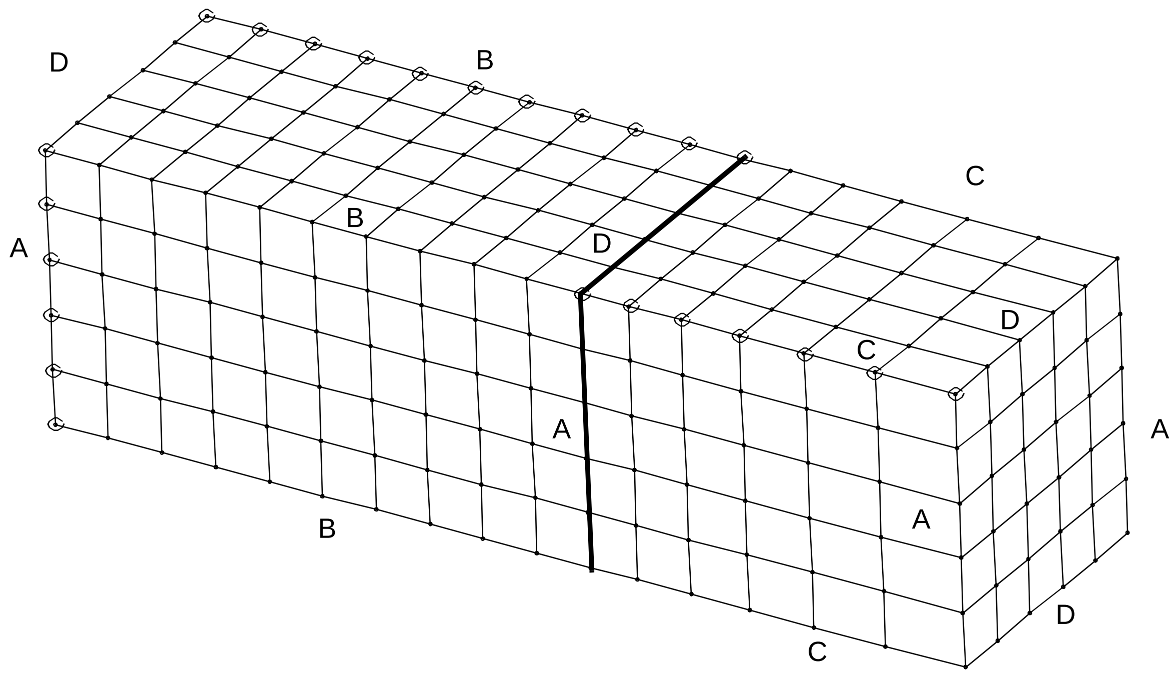


Figure 2-4 Mesh of Two Solids With Seeding Defined

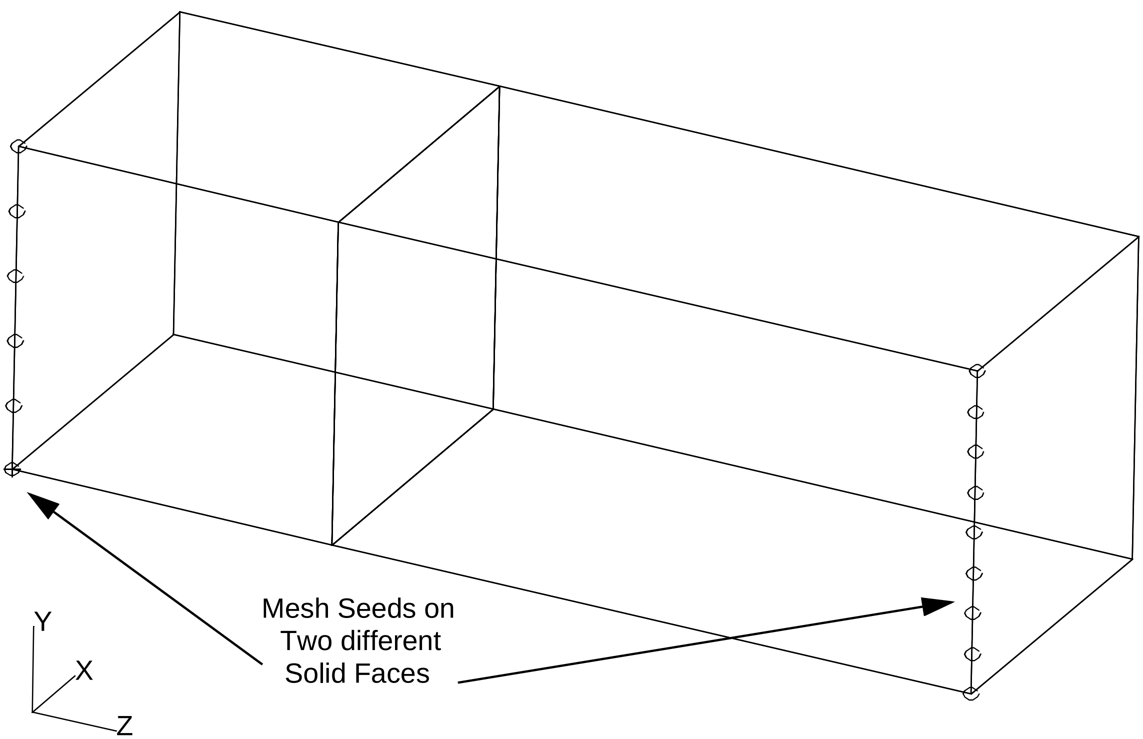


Figure 2-5 Incomplete Mesh Seed Definition for Two Solids

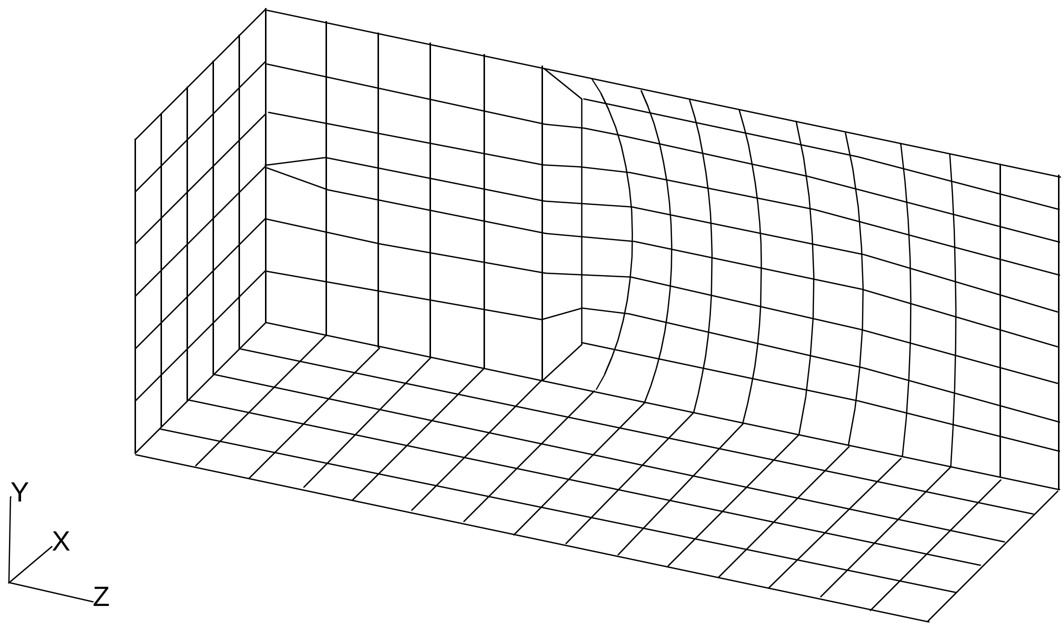


Figure 2-6 Mesh of Two Solids with Additional Mesh Seed

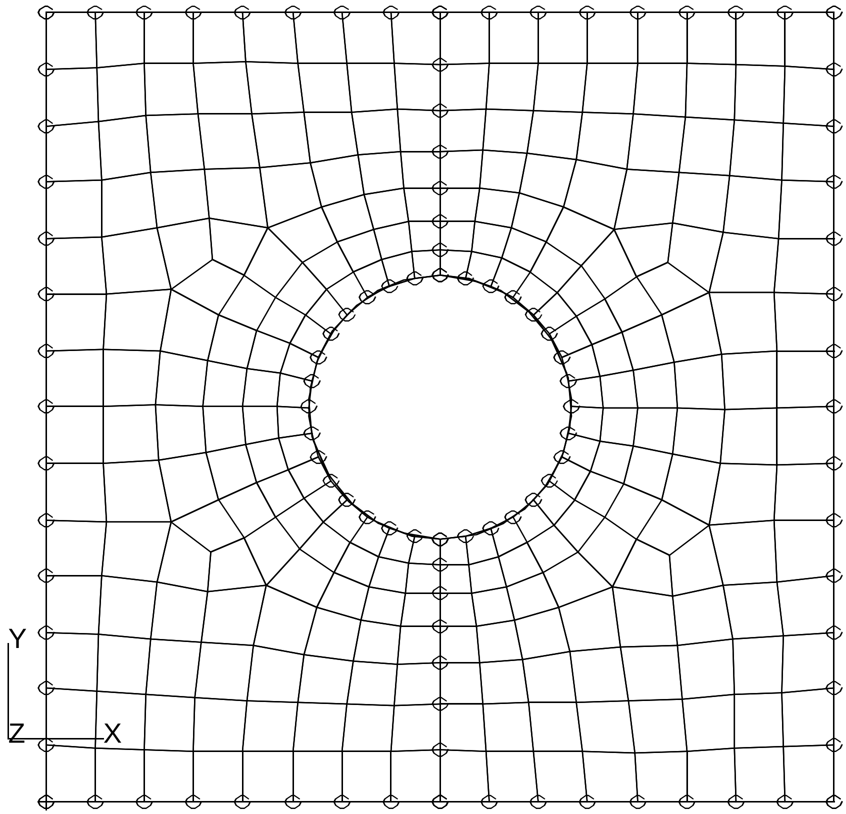


Figure 2-7   Surface Mesh Produced by Paver

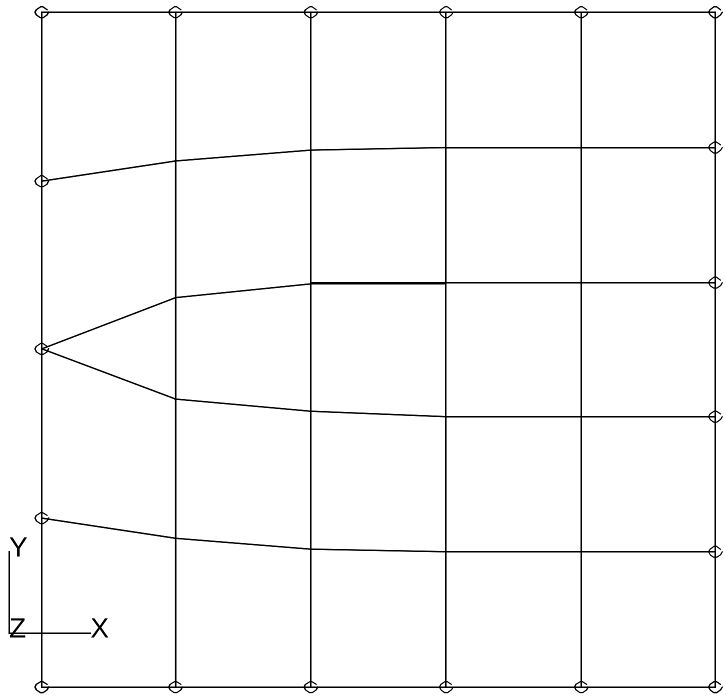


Figure 2-8 Odd Number of Elements Around Surface Perimeter

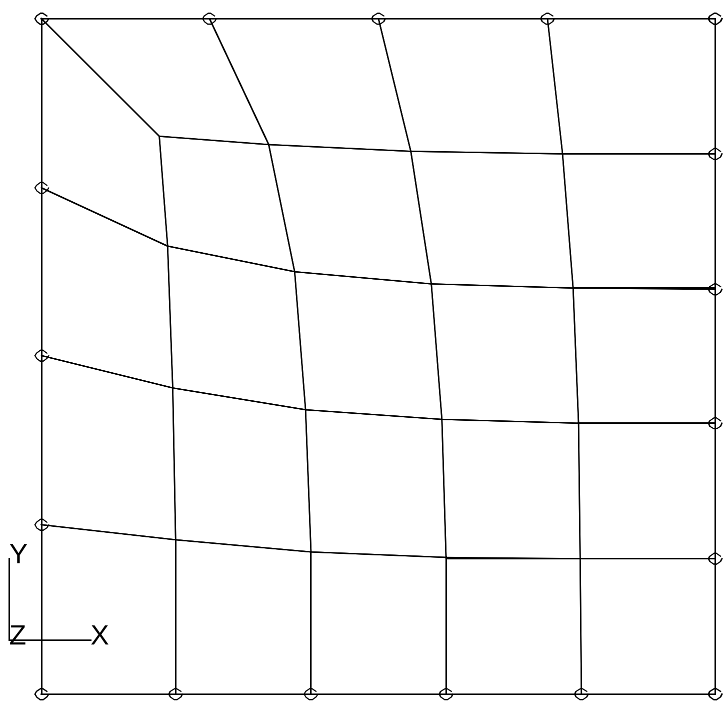


Figure 2-9 Even Number of Elements Around Surface Perimeter

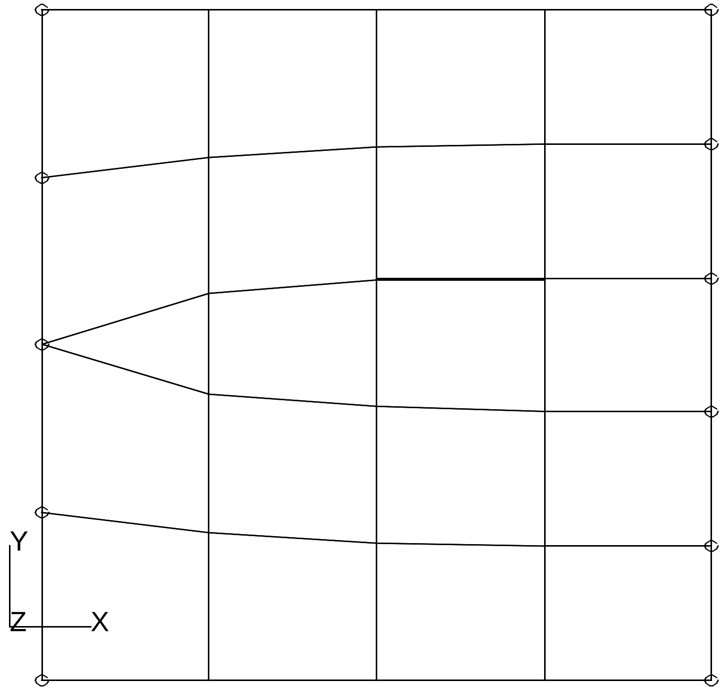


Figure 2-10 Odd Mesh Seed

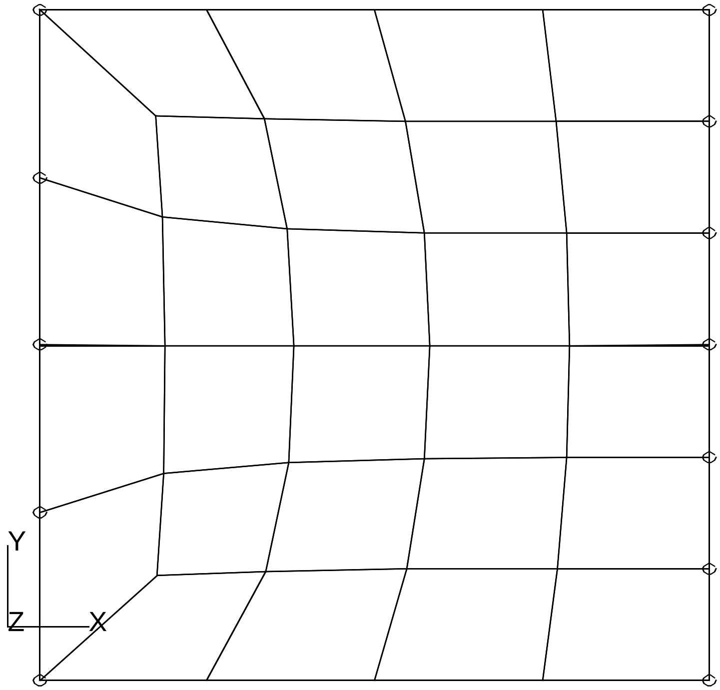


Figure 2-11 Even Mesh Seed

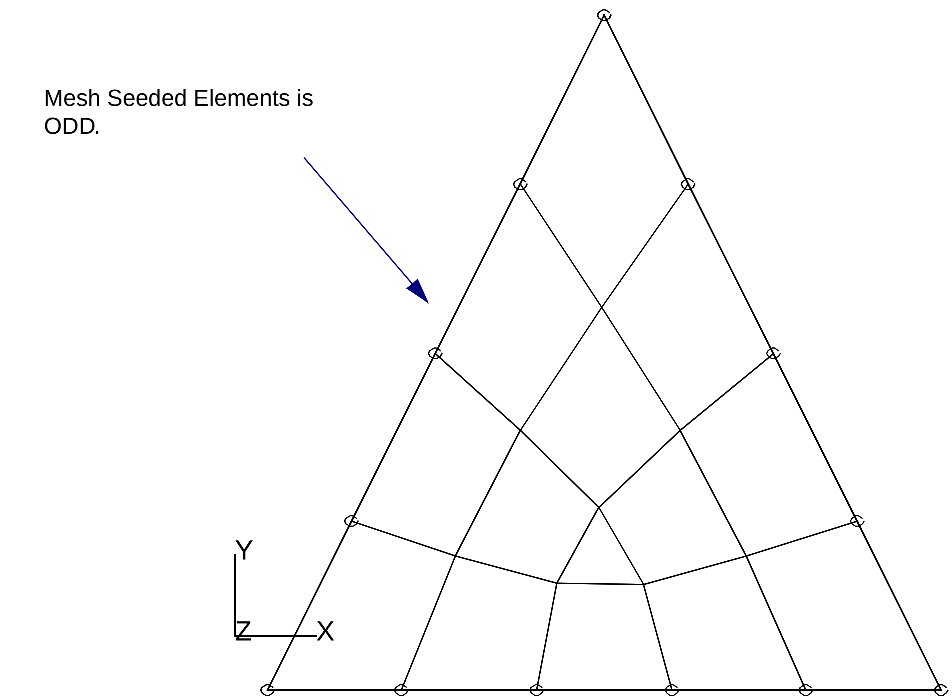


Figure 2-12 Mesh Seeding Triangular Surfaces (1 Tria Produced)

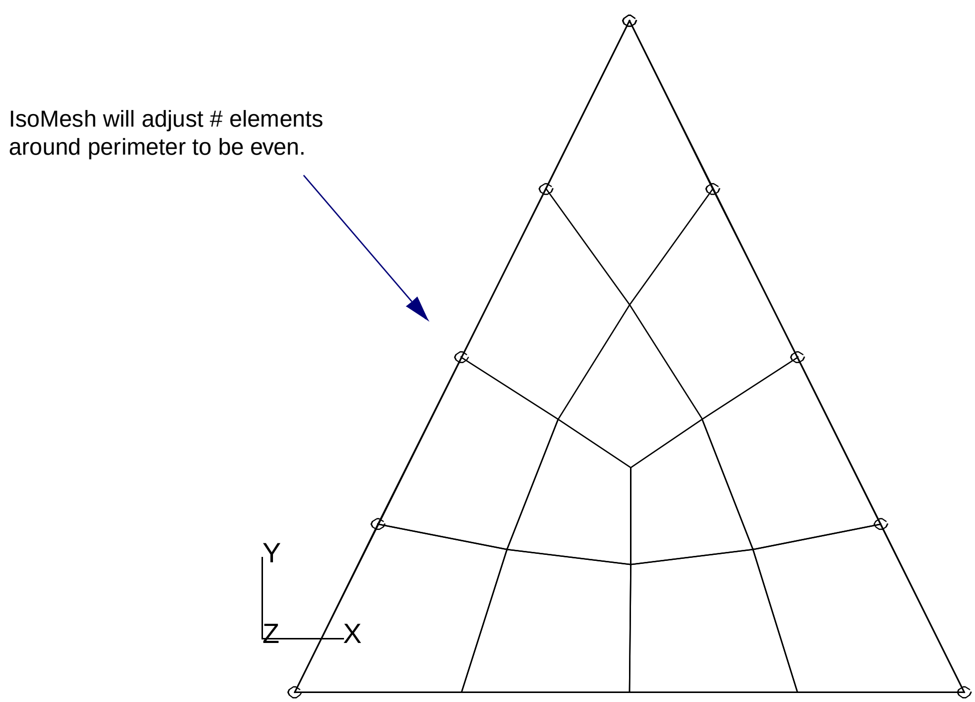


Figure 2-13 Mesh Seeding Triangular Surfaces to Produce only Quad Elements

2.2 Mesh Seed and Mesh Forms

Creating a Mesh Seed

- ❑ Uniform Mesh Seed
- ❑ One Way Bias Mesh Seed
- ❑ Two Way Bias Mesh Seed
- ❑ Curvature Based Mesh Seed
- ❑ Tabular Mesh Seed
- ❑ PCL Function Mesh Seed

Creating a Mesh

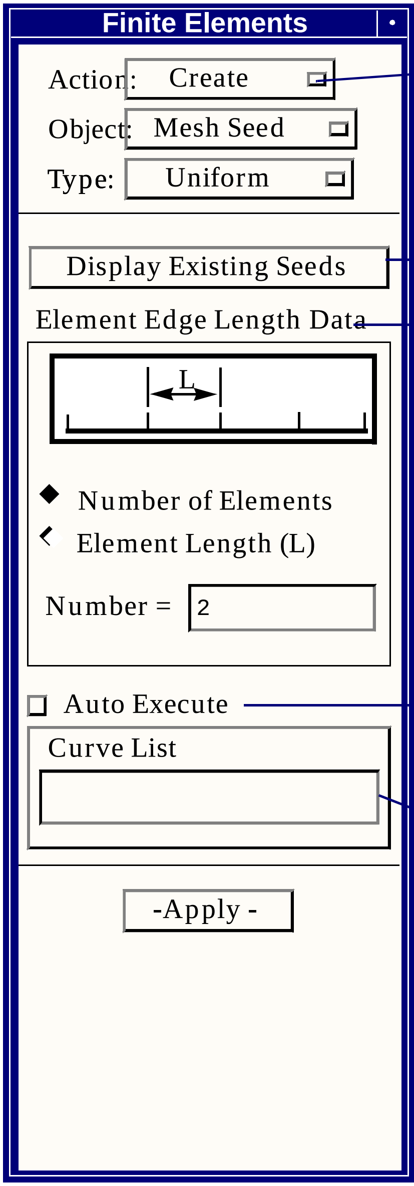
- ❑ IsoMesh Curve
- ❑ IsoMesh 2 Curves
- ❑ IsoMesh Surface
- ❑ Solid
- ❑ Mesh On Mesh
- ❑ Sheet Body
- ❑ Advanced Surface Meshing
- ❑ Auto Hard Points Form

Creating a Mesh Seed

There are many types of mesh seeds: uniform, one way bias, two way bias, curvature based, and tabular.

Uniform Mesh Seed

Create mesh seed definition for a given curve, or an edge of a surface or solid, with a uniform element edge length specified either by a total number of elements or by a general element edge length. The mesh seed will be represented by small yellow circles and displayed only when the Finite Element form is set to creating a Mesh, or creating or deleting a Mesh Seed.



For more information see **One Way Bias Mesh Seed** (p. 31), **Two Way Bias Mesh Seed** (p. 32), **Curvature Based Mesh Seed** (p. 33), **Tabular Mesh Seed** (p. 34) or **PCL Function Mesh Seed** (p. 36).

MSC.Patran will plot all defined mesh seeds associated with the visible geometry.

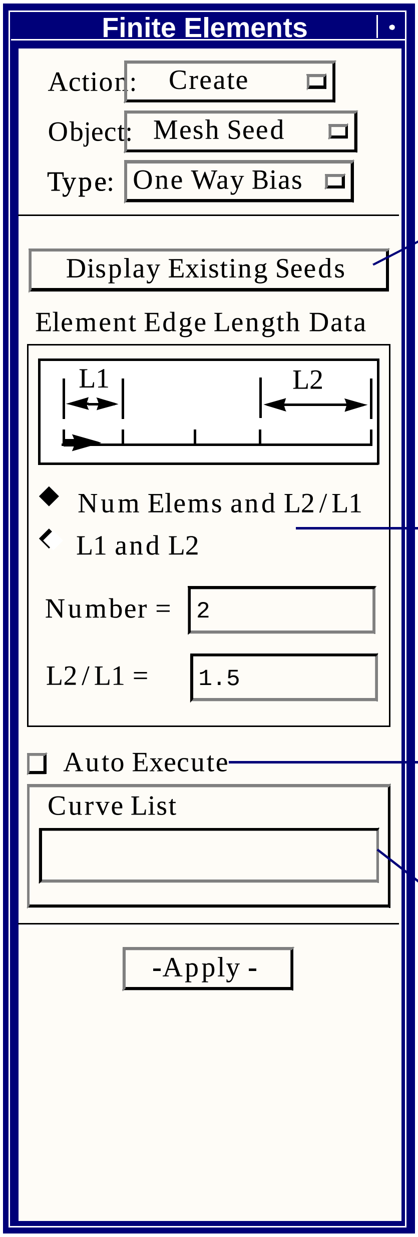
Define node spacing for mesh seed, by either pressing “Number of Elements” or “Element Length (L).” If “Number of Elements” is pressed, the user must then enter an integer value for the desired number of elements. If “Element Length” is pressed, then the user must enter an element edge length (MSC.Patran will calculate the resulting number of elements needed - rounded off to the nearest integer value.

If Auto Execute is selected, MSC.Patran will automatically create a mesh seed definition after each edge is selected. By default Auto Execute is OFF.

Specify list of curves by either cursor selecting existing curves or surface or solid edges, or specifying curve IDs or surface or solid edge IDs. (**Example:** Curve 10, Surface 12.1, Solid 22.5.2.)

One Way Bias Mesh Seed

Create mesh seed definition for a given curve, or an edge of a surface or solid, with an increasing or decreasing element edge length, specified either by a total number of elements with a length ratio, or by actual edge lengths. The mesh seed will be represented by small yellow circles and is displayed only when the Finite Element form is set to creating a Mesh, or creating or deleting a Mesh Seed.



MSC.Patran will plot all defined mesh seeds associated with the visible geometry.

Define node spacing for mesh seed, by either pressing “Num Elems and L2⁄L1” or “L1 and L2”.

If “Num Elems and L2⁄L1” is pressed, the user must enter an integer value for the desired number of elements and an edge length ratio as indicated by the diagram. If “L1 and L2” is pressed, the user must enter edge lengths for the first and last elements.

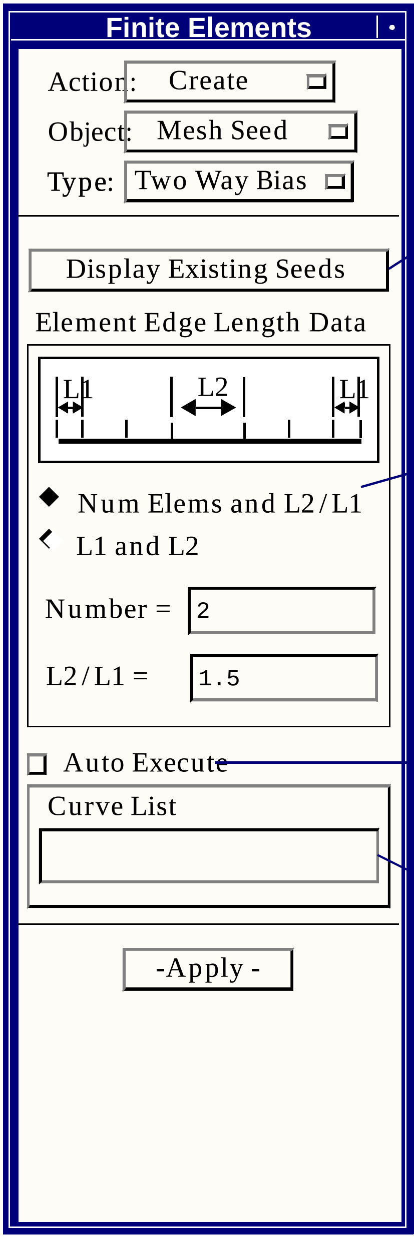
MSC.Patran will calculate the nonuniform mesh seed node spacing through a geometric progression based on the given L2⁄L1 ratio. The positive edge direction for L1 and L2 as indicated by the arrow in the diagram is displayed in the current viewport.

If Auto Execute is selected, MSC.Patran will automatically create a mesh seed definition after each edge is selected. By default Auto Execute is OFF.

Specifies a list of edges by either cursor selecting existing curves or surface or solid edges, or specifying curve IDs or surface or solid edge IDs. (**Example:** Curve 10, Surface 12.1, Solid 22.5.2.)

Two Way Bias Mesh Seed

Create mesh seed definition for a given curve, or an edge of a surface or solid, with a symmetric non-uniform element edge length, specified either by a total number of elements with a length ratio, or by actual edge lengths. The mesh seed will be represented by small yellow circles and is displayed only when the Finite Element form is set to creating a Mesh, or creating or deleting a Mesh Seed.



MSC.Patran will plot all defined mesh seeds associated with the visible geometry.

Define node spacing for mesh seed, by either pressing “Num Elems and L2/L1” or “L1 and L2”.

If “Num Elems and L2/L1” is pressed, the user must enter an integer value for the desired number of elements and an edge length ratio as indicated by the diagram. If “L1 and L2” is pressed, the user must enter edge lengths for the end and middle elements.

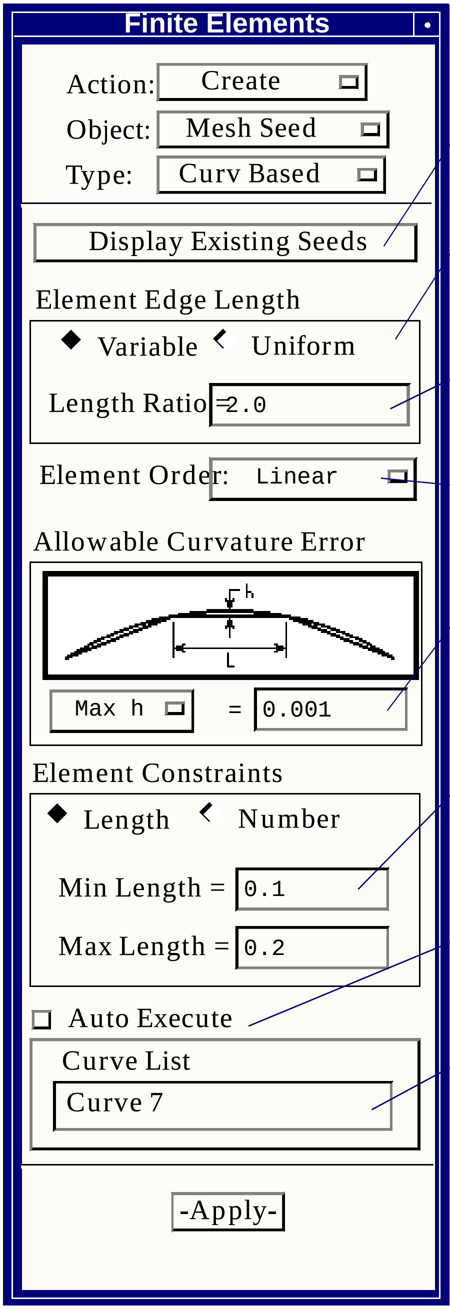
MSC.Patran will calculate the nonuniform mesh seed node spacing through a geometric progression based on the given L2/L1 ratio.

If Auto Execute is selected, MSC.Patran will automatically create a mesh seed definition after each edge is selected. By default Auto Execute is OFF.

Specifies a list of edges by either cursor selecting existing curves or surface or solid edges, or specifying curve IDs or surface or solid edge IDs. (**Example:** Curve 10, Surface 12.1, Solid 22.5.2.)

Curvature Based Mesh Seed

Create mesh seed definition for a given curve, or an edge of a surface or solid, with a uniform or nonuniform element edge length controlled by curvature. The mesh seed will be represented by small yellow circles and is displayed only when the Finite Element form is set to creating a Mesh, or creating or deleting a Mesh Seed.



MSC.Patran will plot all defined mesh seeds associated with the visible geometry.

If Uniform is selected, all elements will have the same length. If Variable is selected, the lengths of elements will vary from longer in regions of gradual curvature to shorter in regions of sharp curvature.

Specify the maximum allowable element edge length ratio. For example a value of 2.0 means that no element may be longer than twice the length of its neighbor.

Select the element order: linear, quadratic or cubic.

Specify the desired maximum curvature error h. (May optionally specify h/L for linear elements.) Number of elements and their lengths will be computed within specified Element Constraints to satisfy Allowable Curvature Error if possible.

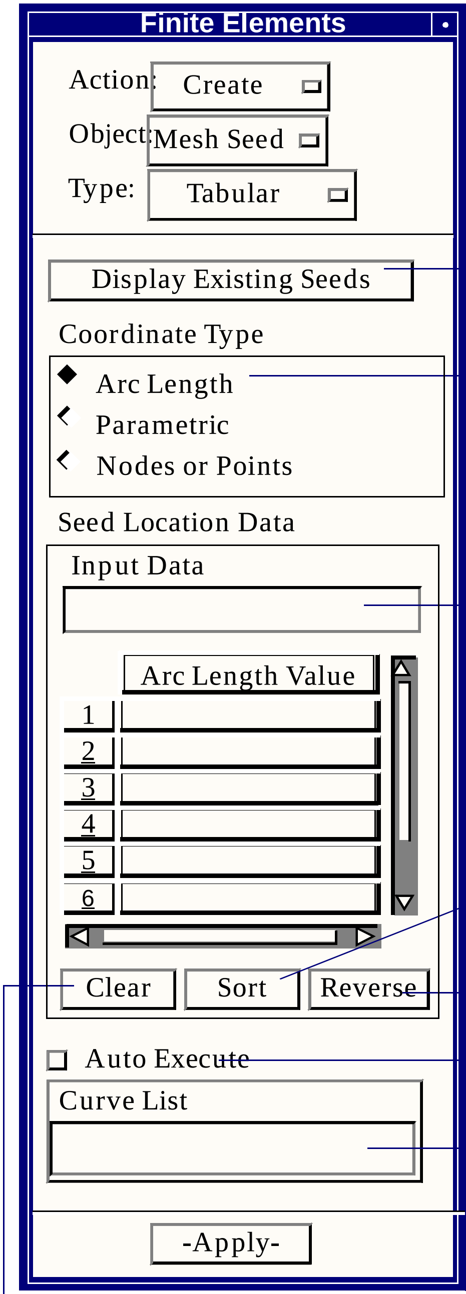
Specify the minimum and maximum number of elements or element lengths to be allowed in constructing the mesh seed. Minimum and maximum number of elements may be the same value. Maximum length must be larger than minimum length.

By default, Auto Execute is turned OFF. This means MSC.Patran will not automatically create a mesh seed definition after each edge is selected.

Specifies a list of edges by either cursor selecting existing curves or surface or solid edges, or specifying curve IDs or surface or solid edge IDs. (**Example:** Curve 10, Surface 12.1, Solid 22.5.2.)

Tabular Mesh Seed

Create mesh seed definition for a given curve, or an edge of a surface or solid, with an arbitrary distribution of seed locations defined by tabular values. The mesh seed will be represented by small yellow circles and is displayed only when the Finite Element form is set to creating a Mesh, or creating or deleting a Mesh Seed.



MSC.Patran will plot all defined mesh seeds associated with all visible geometry.

Specify the coordinate type for the node locations. For example, if Arc length is selected, 0.5 will be located at the mid point of the curve. If Parametric is selected, 0.5 will be located at u=0.5 along the curve.

See next page for Nodes or Points option

Enter the desired node location values. Values can range between 0.0 and 1.0. The values 0.0 and 1.0 will be automatically added if they are omitted.

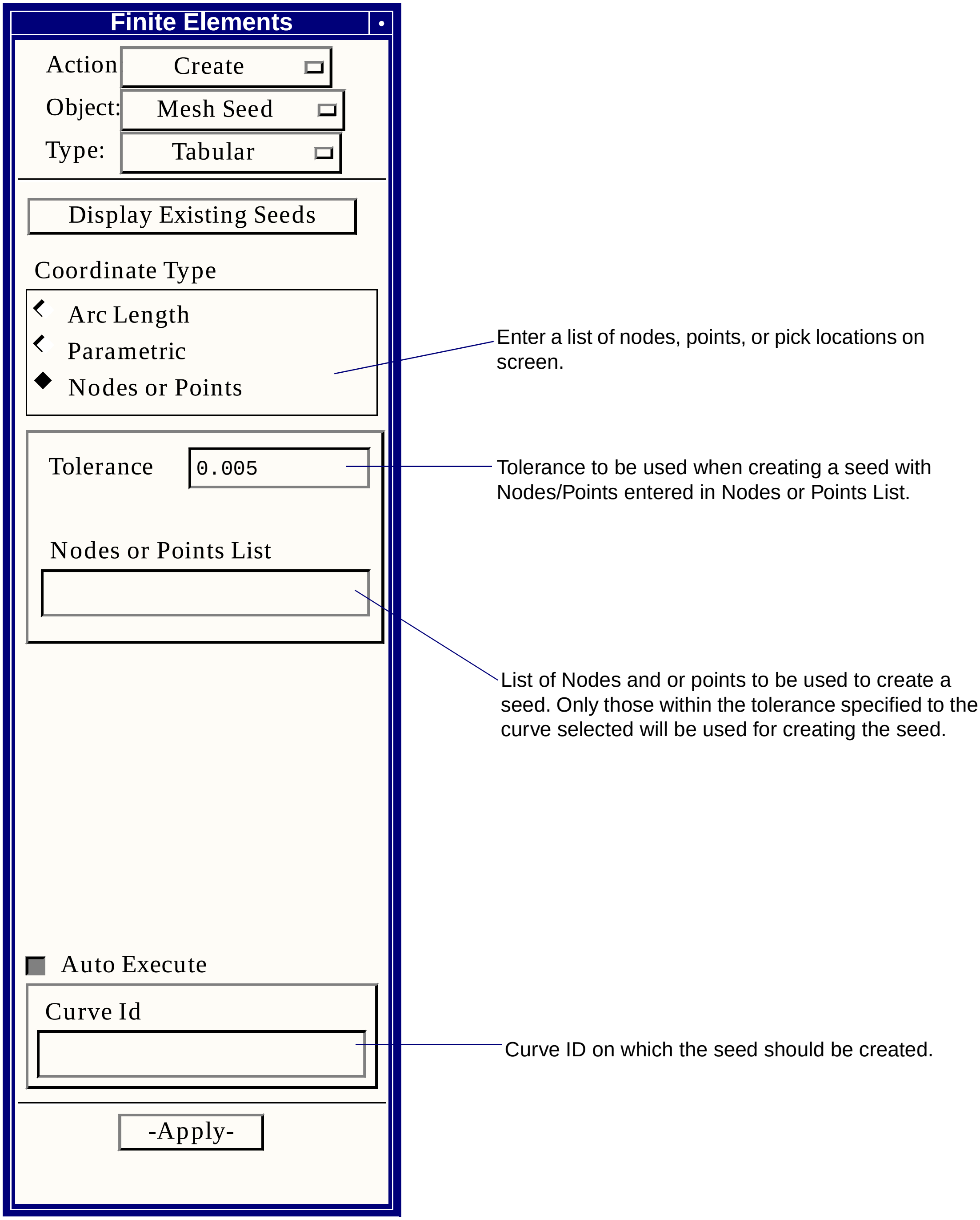
Sorts all the values entered in ascending order.

Reverses the node locations by replacing *v* with 1.0 - *v*.

By default Auto Execute is turned OFF.

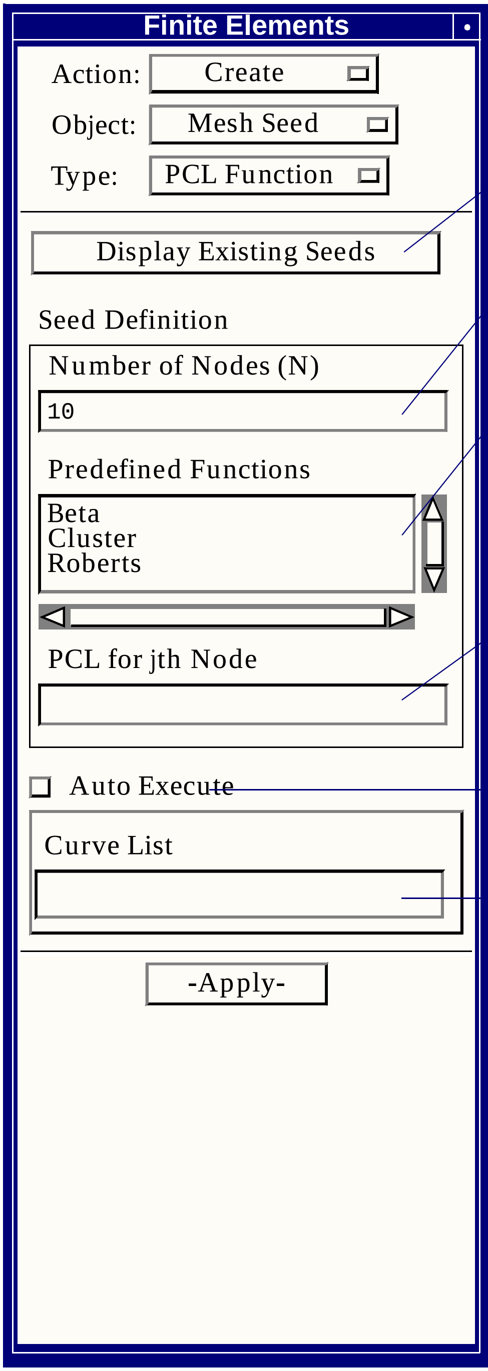
Specify a list of curves or edges of surfaces or solids to which the mesh seeds should be applied. (**Example:** Curve 10, Surface 1.4, Solid 1.8.5.)

Blanks out all cells.


Finite Elements
Action: Create
Object: Mesh Seed
Type: Tabular
Display Existing Seeds
Coordinate Type
Arc Length
Parametric
Nodes or Points
Tolerance
0.005
Nodes or Points List
Auto Execute
Curve Id
-Apply-
Enter a list of nodes, points, or pick locations on screen.
Tolerance to be used when creating a seed with Nodes/Points entered in Nodes or Points List.
List of Nodes and or points to be used to create a seed. Only those within the tolerance specified to the curve selected will be used for creating the seed.
Curve ID on which the seed should be created.

PCL Function Mesh Seed

Create mesh seed definition for a given curve, or an edge of a surface or solid, with a distribution of seed locations defined by a PCL function. The mesh seed will be represented by small yellow circles and is displayed only when the Finite Element form is set to creating a Mesh, or creating or deleting a Mesh Seed.



MSC.Patran will plot all defined mesh seeds associated with all visible geometry.

Enter the number of nodes (seeds) to be created.

Selecting one of the predefined functions will enter its call into the “PCL for jth Node” data box where it can be edited if desired. These functions exist in the PCL library supplied with MSC.Patran. Sample code is presented on the next page for use as a model in writing your own function.

Enter an inline PCL function or a call to an existing compiled function. A trivial example of an inline function is one that computes a unifor seed:

(j-1)/N-1)

By default Auto Execute is turned OFF.

Specify a list of curves or edges of surfaces or solids to which the mesh seeds should be applied.

(**Example:** Curve 10, Surface 1.4, Solid 22.5.3)

The following is the PCL code for the predefined function called beta. It may be used as a model for writing your own PCL function.

```
FUNCTION beta(j, N, b)
GLOBAL INTEGERj, N
REALb, w, t, rval

x = (N - j) / (N - 1)
t = (   ( b  +  1.0 )     /   ( b  -  1.0 )     ) **w
rval = (  (b + 1.0)  -  (b - 1.0) *t  )  /  (t + 1.0)
RETURN rval
END FUNCTION
```

```
Note: j and N MUST be the names for the first two arguments.
```

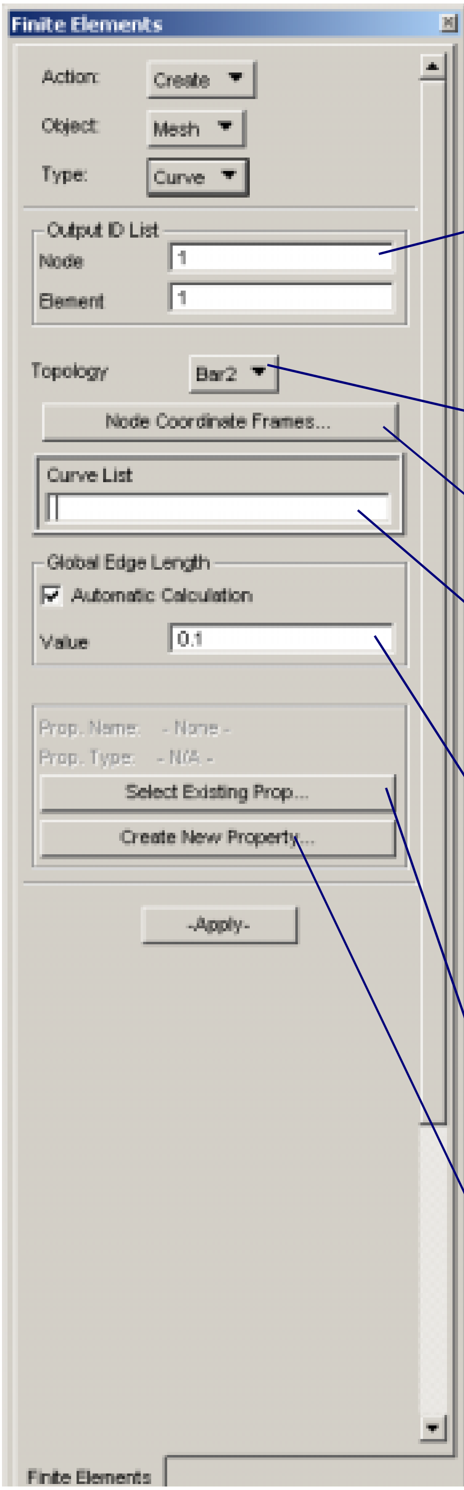
```
N is the number of nodes to be created, and j is the index of the node being
created, where ( 1 <= j <= N ).
```

```
An individual user function can be accessed at run time by entering the command:

!!INPUT <my_pcl_function_file_name>

A library of precompiled PCL functions can be accessed by:

!!LIBRARY <my_plb_library_name>

For convenience these commandes can be entered into your p3epilog.pcl file so
that the functions are available whenever you run MSC.Patran.
```

2.3 Creating a Mesh

There are several geometry types from which to create a mesh:

IsoMesh Curve



Assigns an optional list of ID numbers for a new set of nodes and elements. If not specified, ID values will be assigned consecutively starting with the node and element ID shown.

ID lists containing duplicate IDs, or IDs of preexisting nodes or elements will be rejected. Users must first delete the existing node or element with the specified ID before reusing the same ID in a later list.

The maximum ID limit for nodes or elements is approximately 2 billion (2^{31}-1). The only limit to the number of nodes and elements which can be created per geometric entity is the amount of available disk space.

Choose the type of bar element to create from the pop-up list. Available bar elements to choose from are Bar2, Bar3, and Bar4.

Brings up the Node Coordinate Frames form. This allows an Analysis and a Reference Coordinate system to be defined for the next mesh of nodes.

Specify list of curves to mesh by either cursor selecting existing curves, surface edges, or solid edges, or by specifying the curve IDs, or the surface or solid edge IDs. (**Example:** Curve 12, Surface 1.3, Solid 11.3.1.)

Specify a real value to assign the default element edge length for a given mesh. This value will not override any predefined mesh seeded edges. Global edge lengths will only be applied where mesh seeds have not been defined.

The default Global Edge Length value can be set using the pref_env_set_real function with argumet “DefaultMeshSize” in the settings.pcl file. When defined in this way, the initial activation of the Automatic Calculation toggle is disabled.

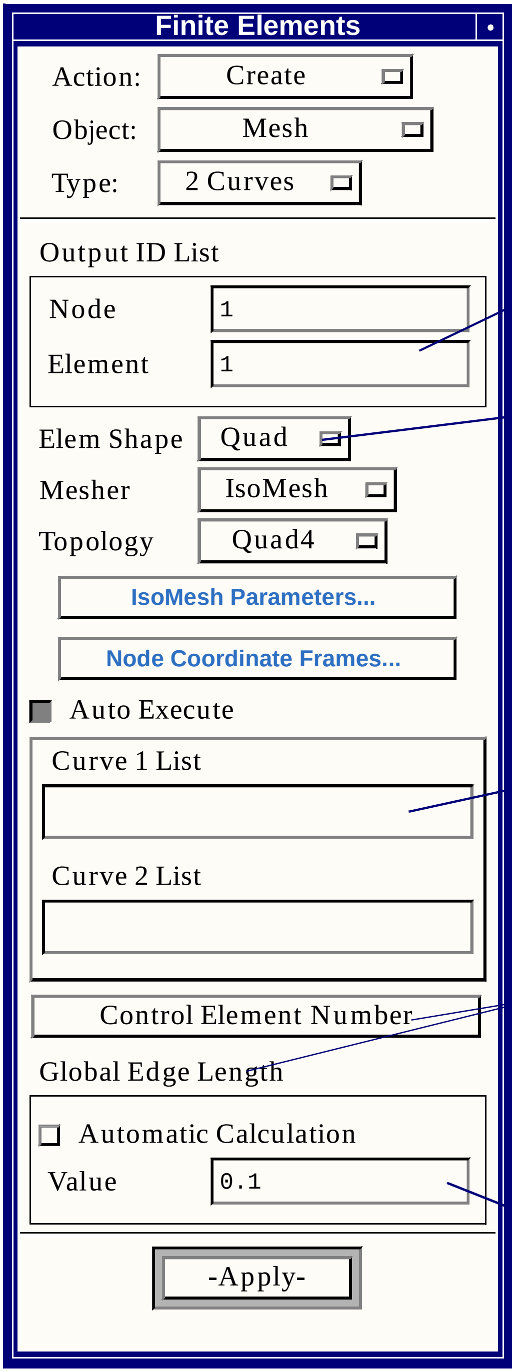
Please see **pref_env_set_real** (p. 1463) in the *PCL Reference Manual, Volume 1: Function Descriptions*

Use this button to select an existing property to assign to newly created elements. Once a property is selected it's name will appear in the "Prop. Name:" label, and a small description of that property type will appear in the "Prop. Type." label.

Use this button to create a property to assign to the newly created elements. Once the property is created, it is automatically selected as the property to associate to the new elements.

Don’t forget to reset the Global Edge Length to the appropriate value before applying the mesh.

IsoMesh 2 Curves



Assigns an optional list of ID numbers for a new set of nodes and elements. If not specified, ID values will be assigned consecutively starting with the node and element ID shown.

ID lists containing duplicate IDs, or IDs of preexisting nodes or elements will be rejected. Users must first delete the existing node or element with the specified ID before reusing the same ID in a later list.

The maximum ID limit for nodes or elements is approximately 2 billion (2^{31}-1). The only limit to the number of nodes and elements which can be created per geometric entity is the amount of available disk space.

Choose the type of Quad or Tria element to create from the given list. Available Quad and Tria elements to choose from are Quad4, Quad5, Quad8, Quad9, Quad12, Quad16, Tria3, Tria4, Tria6, Tria7, Tria9, Tria13.

Specify the two curve lists. Isomesh with selected element type will be created on a ruled surface connecting the two input curve lists.

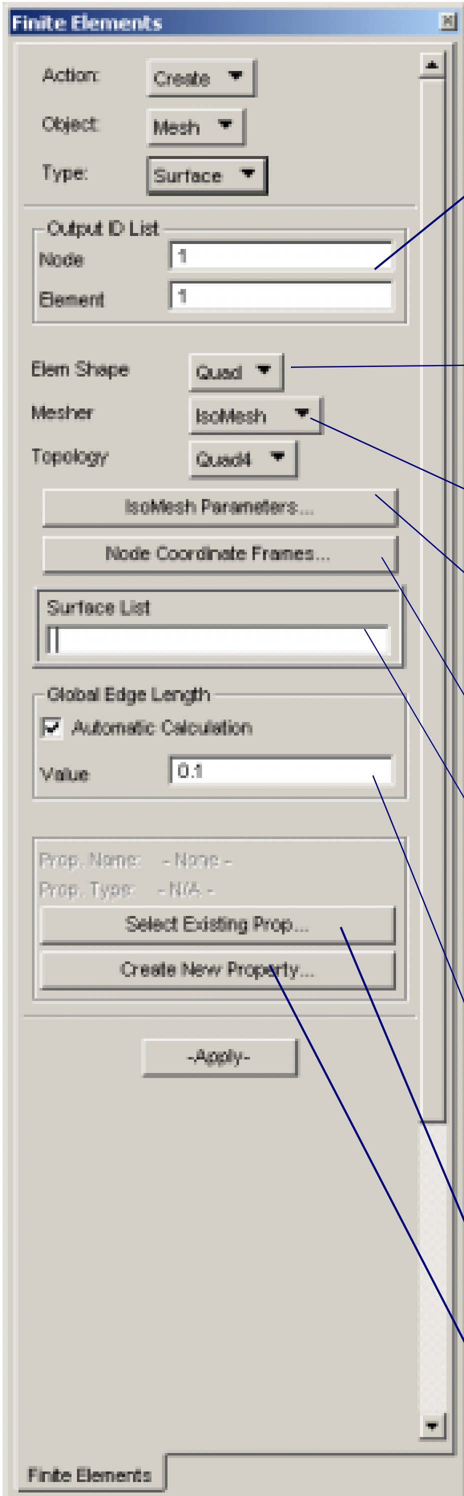
Specify number of elements on edges or a real value to assign the default element edge length for a given mesh.

Both options will not override any predefined mesh seeded edges. Global edge lengths and element number will only be applied where mesh seeds have not been defined.

The default Global Edge Length value can be set using the pref_env_set_real function with argument “DefaultMeshSize” in the settings.pcl file. When defined in this way, the initial activation of the Automatic Calculation toggle is disabled.

Please see **pref_env_set_real** (p. 1463) in the *PCL Reference Manual, Volume 1: Function Descriptions*

IsoMesh Surface



Don't forget to reset the Global Edge Length to the appropriate value before applying the mesh.

Assigns an optional list of ID numbers for a new set of nodes and elements. If not specified, ID values will be assigned consecutively starting with the node and element ID shown.

ID lists containing duplicate IDs, or IDs of preexisting nodes or elements will be rejected. Users must first delete the existing node or element with the specified ID before reusing the same ID in a later list.

The maximum ID limit for nodes or elements is approximately 2 billion (2^{31}-1). The only limit to the number of nodes and elements which can be created per geometric entity is the amount of available disk space.

Choose the type of Quad or Tria element to create from the given list. Available Quad and Tria elements to choose from are Quad4, Quad5, Quad8, Quad9, Quad12, Quad16, Tria3, Tria4, Tria6, Tria7, Tria9, Tria13.

Choose either the IsoMesh method or the Paver method of meshing. If Paver is selected the IsoMesh Parameters changes to **Paver Parameters...**

Brings up the IsoMesh Parameters form which is used for transition meshes. This is an optional function that affects MSC.Patran's IsoMesh smoothing algorithm For most transition meshes, it is not required to reset the default parameter values.

Brings up the Node Coordinate Frames form. This allows an Analysis and a Reference Coordinate system to be defined for the next mesh of nodes.

Specifies a list of surfaces to mesh by either cursor selecting existing surfaces, solid faces, or SGM trimmed faces, or entering the IDs of the surfaces, solid faces, and/or SGM trimmed faces. (**Example:** Surface 23, Solid 11.3.)

Specify a real value to assign the default element edge length for a given mesh. This value will not override any predefined mesh seeded edges. Global edge lengths will only be applied where mesh seeds have not been defined or where there are no existing adjacent meshed regions.

The default Global Edge Length value can be set using the pref_env_set_real function with argument "DefaultMeshSize" in the settings.pcl file. When defined in this way, the initial activation of the Automatic Calculation toggle is disabled.

Please see **pref_env_set_real** (p. 1463) in the *PCL Reference Manual, Volume 1: Function Descriptions*

Use this button to select an existing property to assign to newly created elements. Once a property is selected it's name will appear in the "Prop. Name:" label, and a small description of that property type will appear in the "Prop. Type." label.

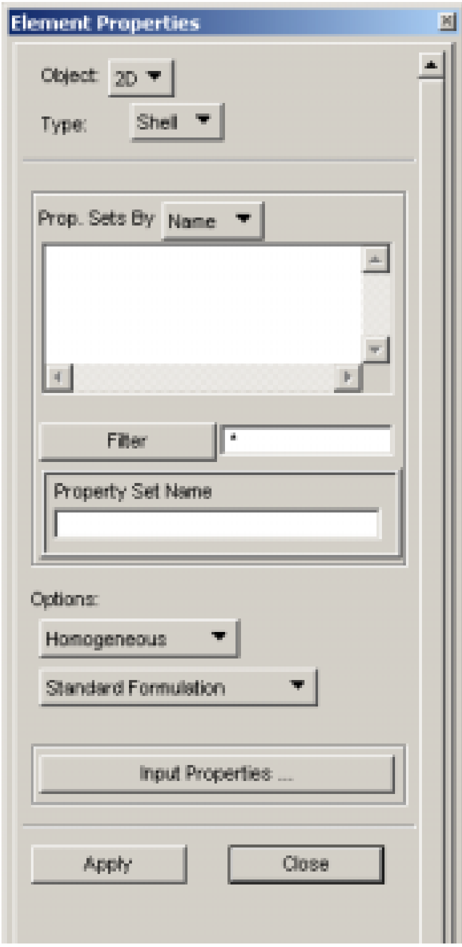
Use this button to create a property to assign to the newly created elements. Once the property is created, it is automatically selected as the property to associate to the new elements.

Property Sets

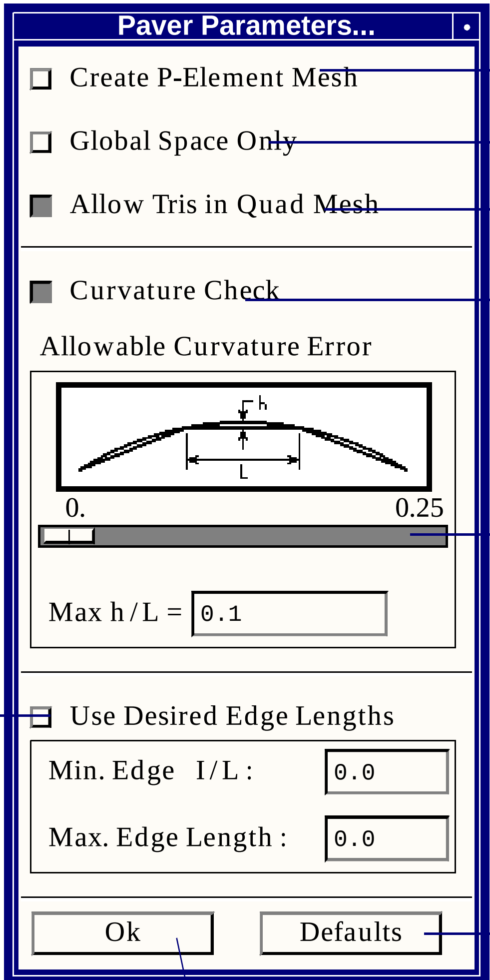
Use this form to select existing Properties to associate with elements to be created.

Create New Property

Use this form to create a property set and associate that property set to the elements being created. This form behaves exactly like the Properties Application Form.



Paver Parameters



Causes the mesher to relax its criteria for element quality in an attempt to create a coarser mesh.

Mesh the surface in global space only, if the toggle is set.

Allows one tri element on a loop if the sum of all the element edges on the loop is an odd number.

Causes the mesher to adjust the mesh density to control the deviation between the geometry and the straight element edges. This will usually result in a better mesh and a better chance for success when meshing complicated geometry. Currently, this only works on edges only.

If **Curvature Check** is set, enter the desired maximum deviation between the element edges and geometry as the ratio of the deviation to element edge length. Deviation is measured at the center of the element edge. The value may be entered either by using the slider bar or by typing a value into the data box.

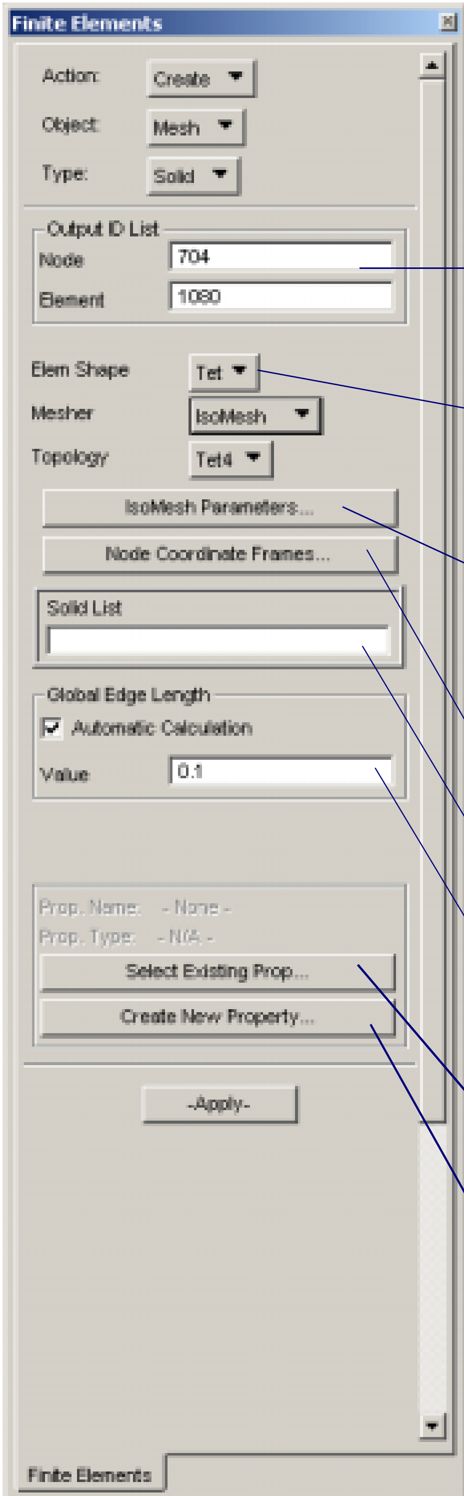
Used to restore all of the default settings to this form.

The **Ok** button must be clicked with the mouse in order to update the new values the user enters on this form and make them available to the meshers. If the button is not clicked, the new values will not be passed on to the meshers.

The Paver has more freedom to adjust the mesh size in the geometry interior. If this toggle is not set, then the meshers will try to make elements in the interior sized between the shortest and longest edge on the boundary, or the global length from the main meshing form, whichever is more extreme. This is how the paver worked in previous MSC.Patran releases. If this toggle is set and reasonable values for minimum and maximum element edge lengths are given, then the meshers will attempt to make elements in the interior of the meshing region with element edge lengths between the user defined values. This gives the user more control over the mesh in the interior of the geometry.

Solid

IsoMesh



Assigns an optional list of ID numbers for a new set of nodes and elements. If not specified, ID values will be assigned consecutively starting with the node and element ID shown.

ID lists containing duplicate IDs, or IDs of preexisting nodes or elements will be rejected. Users must first delete the existing node or element with the specified ID before reusing the same ID in a later list.

The maximum ID limit for nodes or elements is approximately 2 billion (2^{31}- 1). The only limit to the number of nodes and elements that can be created per geometric entity is the amount of available disk space.

Choose the type of Hex, Wedge or Tet element to create from the given list. Available solid elements to choose from are Hex8, Hex9, Hex20, Hex21, Hex26, Hex27, Hex32, Hex64, Wedge6, Wedge7, Wedge15, Wedge16, Wedge20,Wedge21, Wedge24, Wedge52, Tet4, Tet5, Tet10, Tet11, Tet14, Tet15, Tet16, Tet40.

Brings up the IsoMesh Parameters form which is used for transition meshes. This is an optional function that affects MSC.Patran’s IsoMesh smoothing algorithm. For most transition meshes, it is not required to reset the default parameter values. If TetMesh is selected the IsoMesh Parameters changes to **TetMesh Parameters...**

Brings up the Node Coordinate Frames form which allows an Analysis and a Reference Coordinate system to be defined for the next mesh of nodes.

Specifies a list of solids to mesh by either cursor selecting existing solids, or entering the IDs of the solids. (**Example:** Solid 23.)

Specify a real value to assign the default element edge length for a given mesh. This value will not override any predefined mesh seeded edges. Global edge lengths will only be applied where mesh seeds have not been defined or where there are no existing adjacent meshed regions.

Use this button to select an existing property to assign to newly created elements. Once a property is selected it's name will appear in the "Prop. Name:" label, and a small description of that property type will appear in the "Prop. Type." label.

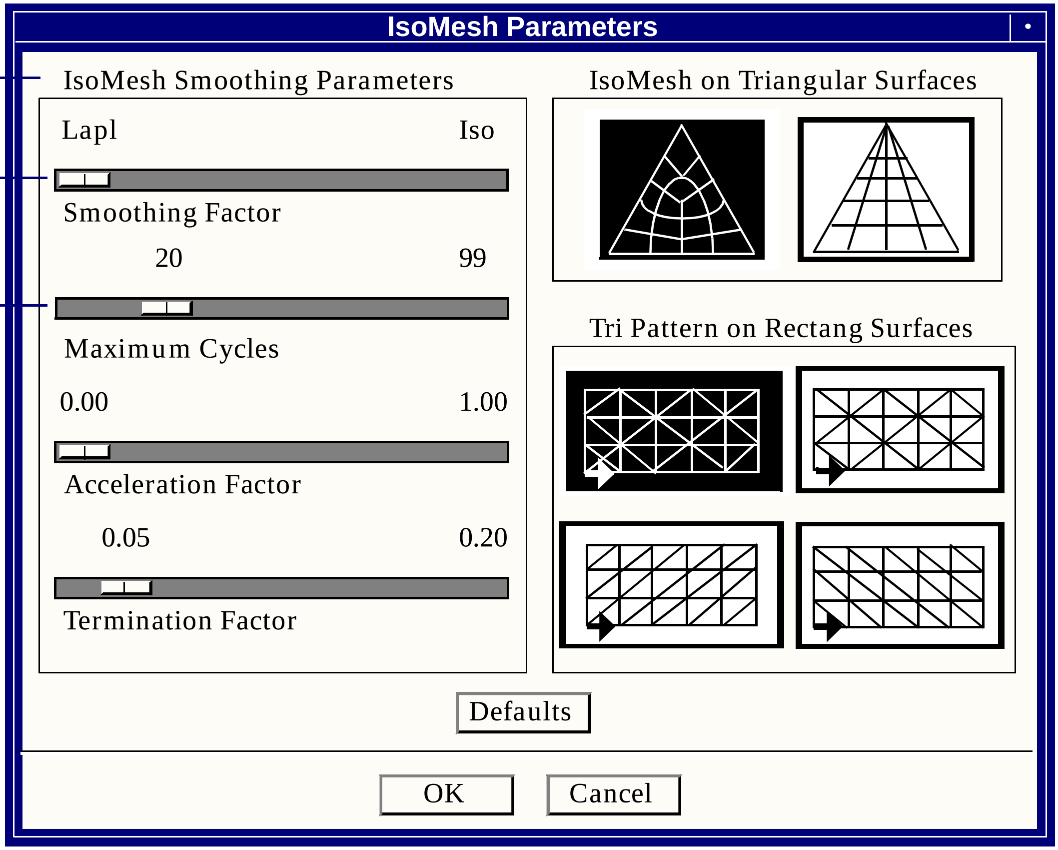
Use this button to create a property to assign to the newly created elements. Once the property is created, it is automatically selected as the property to associate to the new elements.

Don’t forget to reset the Global Edge Length to the appropriate value before applying the mesh.

IsoMesh Parameters Subordinate Form. This form appears when the IsoMesh Parameters button is selected on the Finite Elements form.

Smoothing parameters affect only transition meshes. A transition occurs when two opposing edges of a surface differ in the number of elements or mesh ratio. The values may be changed by pressing the left mouse button and moving the slide bar to the appropriate value.

The smoothing algorithm used by MSC.Patran is the iterative Laplacian-Isoparametric scheme developed by L.R. Herrmann.

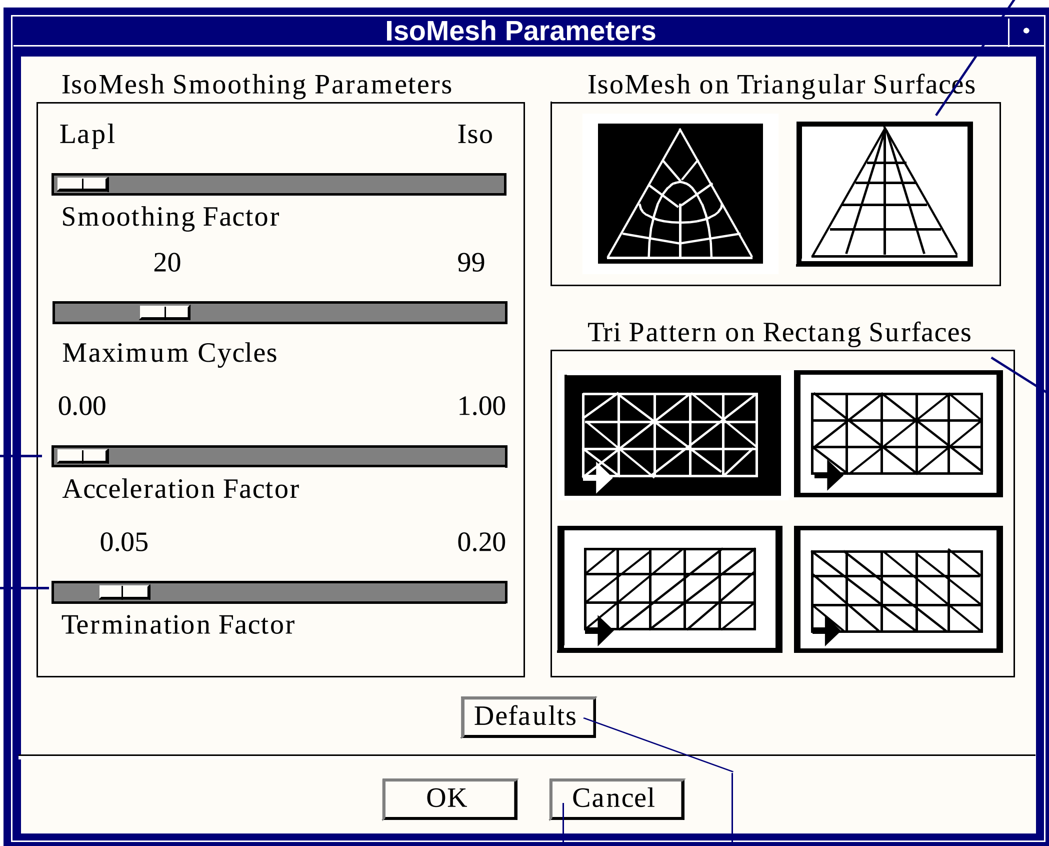


Maximum number of iterations allowed for mesh smoothing. Default value is 20. Smoothing may be turned off by setting the Maximum Cycles to zero.

Used to determine a weighted combination of Laplacian and Isoparametric smoothing methods. Valid range is from 0.0 to 1.0, where 0.0 is pure Laplacian smoothing and 1.0 is pure Isoparametric smoothing. Intermediate values mean a combination of the two methods will be used. The default value is 0.0. Laplacian smoothing is best for most transition cases, except where the surface has significant inplane curvature in that case, Isoparametric smoothing is best.

Used to accelerate the mesh smoothing. The default value is 0.0. A value of 0.3 to 0.5 may cause mesh smoothing to converge in fewer cycles. For example: a value of 0.5 would cause each node to move 50% farther than computed at each iteration. However, the following warning will be issued if the acceleration factor is reset to a nonzero value: “Nonzero acceleration factor may cause mesh smoothing failure if geometry is highly curved or skewed”.

Choose from two available mesh patterns for degenerate surfaces or solids.



Choose from four available triangular mesh patterns for surfaces or solids with 90 degree

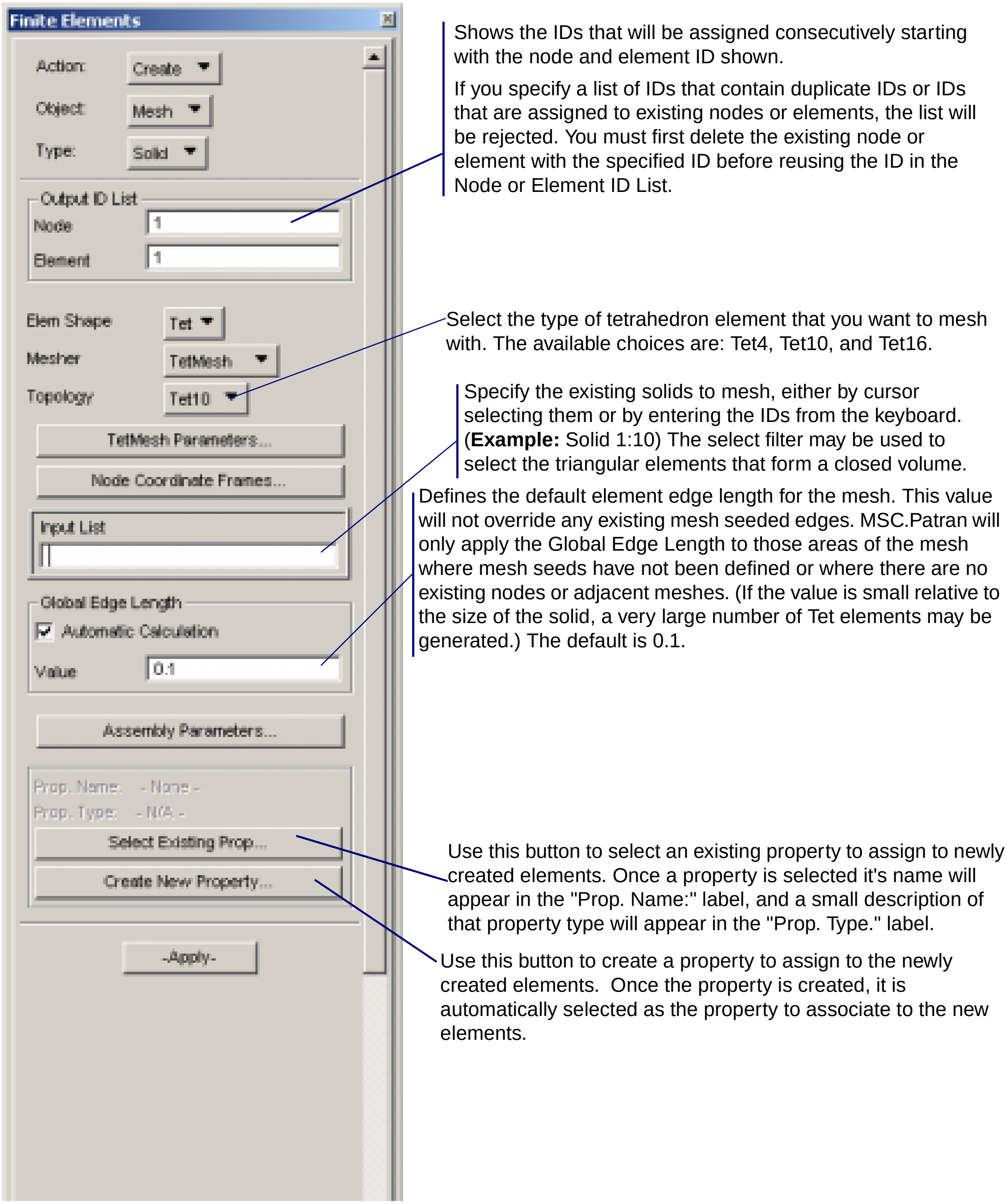
Controls the smoothing termination tolerance factor. Default value is 0.05. It stops the mesh smoothing when the largest distance moved by any node during the previous iteration is less than 5% of the shortest element length along an edge of the geometry.

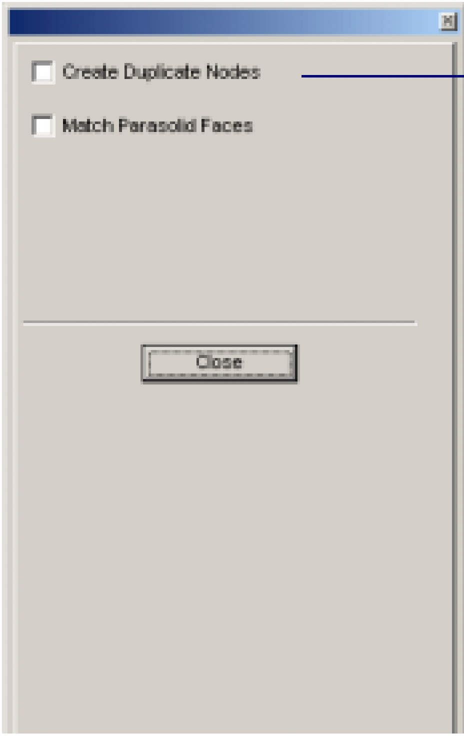
If pressed, the smoothing parameters will be reset back to the original “factory” default values. These are: Smoothing Factor = 0.0, Maximum Cycles = 20, Acceleration Factor = 0.00, Termination Factor = 0.05.

If selected, MSC.Patran will reset the smoothing parameter values back to the original set of values that existed upon entry to the Mesh Parameter form.

TetMesh

Using the **Create/Mesh/Solid** form with the **TetMesh** button pressed creates a set of four node, 10 node or 16 node tetrahedron elements for a specified set of solids. The solids can be composed of any number of sides or faces.





The duplicate nodes toggle will create an extra set of nodes on geometry that is shared between more than one solid, for example on a face that is shared between two solids.

This toggle also applies to assembly meshing, for example, using Match Parasolid Faces, where a duplicate set of nodes will be created on entities that are found to be shared between solids. The only situation in assembly meshing where duplicate nodes will be created with the toggle off is where an existing mesh is being transferred from a neighboring solid.

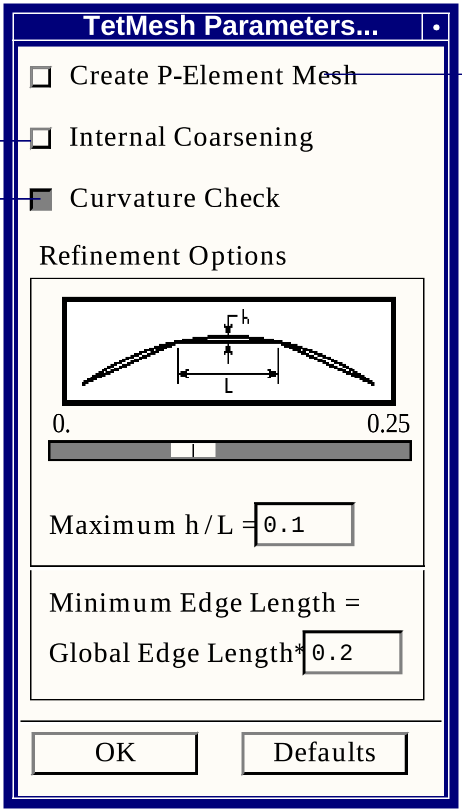
More Help:

- **Creating a Boundary Representation (B-rep) Solid** (p. 338) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*
- **Solids** (p. 24) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*
- Verify - Element (Normals) (p. 196)

TetMesh Parameters

The TetMesh Parameters sub-form allows you to change meshing parameters for P-Element meshing and Curvature based refinement.

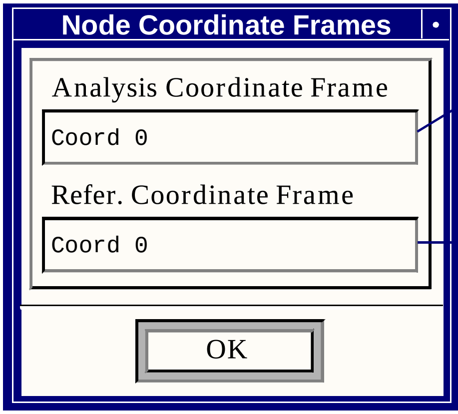
The tetrahedral mesh generator has an option to allow for transition of the mesh from a very small size to the user given Global Edge Length. This option can be invoked by turning the Internal Coarsening toggle ON. This option is supported only when a solid is selected for meshing. The internal grading is governed by a growth factor, which is same as that used for grading the surface meshes in areas of high curvature (1:1.5). The elements are gradually stretched using the grade factor until it reaches the user given Global Edge Length. After reaching the Global Edge Length the mesh size remains constant.



When creating a mesh with mid-side nodes (such as with Tet10 elements) in a solid with curved faces, it is possible to create elements that have a negative Jacobian ratio which is unacceptable to finite element solvers. To prevent an error from occurring during downstream solution pre-processing, the edges for these negative Jacobian elements are automatically straightened resulting in a positive Jacobian element. Although the solver will accept this element's Jacobian, the element edge is a straight line and no longer conforms to the original curved geometry. If this toggle is enabled before the meshing process, the element edges causing a negative Jacobian will conform to the geometry, but will be invalid elements for most solvers. To preserve edge conformance to the geometry, the "Modify-Mesh-Solid" functionality can then be utilized to locally remesh the elements near the elements containing a negative Jacobian.

To create a finer mesh in regions of high curvature, the "Curvature Check" toggle should be turned ON. There are two options to control the refinement parameters. Reducing the "Maximum h/L" creates more elements in regions of high curvature to lower the distance between the geometry and the element edge. The "Minimum l/L" option controls the lower limit of how small the element size can be reduced in curved regions. The ratio l/L is the size of the minimum refined element edge to the "Global Edge Length" specified on the "Create-Mesh-Solid" form.

Node Coordinate Frames



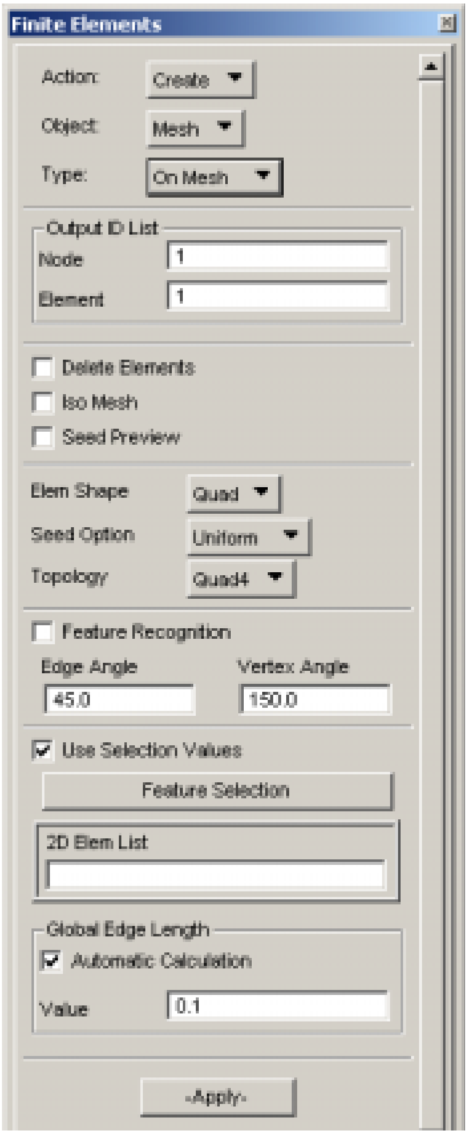
Specifies local coordinate frame ID for analysis results. The default coordinate frame ID is defined under the Global Preferences menu, usually the global rectangular frame ID of zero.

Specifies an existing local coordinate frame ID to associate with the set of meshed nodes. The nodes' locations can later be shown or modified within the specified reference frame. See **The Show Action** (Ch. 12). The default coordinate frame ID is defined under the Global Preferences menu, usually the global rectangular frame ID of zero.

Mesh On Mesh

Mesh On Mesh is a fem-based shell mesh generation program. It takes a shell mesh as input, and creates a new tria / quad mesh according to given mesh parameters. It works well even on rough tria-meshes with very bad triangles created from complex models and graphic tessellations (STL data). Mesh On Mesh is also a re-meshing tool. You can use it to re-mesh a patch on an existing mesh with a different element size.

This mesher has two useful features: feature recognition and preservation, and iso-meshing. If the feature recognition flag is on, the ridge features on the input mesh will be identified based on the feature edge angle and vertex angle, and will be preserved in the output mesh. Also, Mesh On Mesh will recognize 4-sided regions automatically and create good iso-meshes on these regions.



Delete Elements If checked, the input elements will be deleted.

Iso Mesh If checked, an iso-mesh will be created on a 4-sided region. You need to select 4 corner nodes in the data box Feature Selection / Vertex Nodes.

Seed Preview If checked, the mesh seeds on the boundary and feature lines will be displayed as bar elements. The vertices on the boundary and feature lines will be displayed as point elements. This option is useful if users turn on the Feature Recognition Flag and want to preview the feature line setting before creating a mesh.

Element Shape Element Shape consists of:

- Quad
- Tria

Seed Option

- **Uniform.** The mesher will create new boundary nodes based on input global edge length.
- **Existing Boundary.** All boundary edges on input mesh will be preserved.
- **Defined Boundary.** The mesher will use all the nodes selected in the data box Boundary Seeds to define the boundary of the output mesh. No other boundary nodes will be created.

Topology

- Quad4
- Tria3

Feature Recognition If checked, the features on the input mesh will be defined automatically based on feature edge angles and vertex angles, and be preserved on the output mesh.

Vertex Angle If Feature Recognition is on, a node on a feature line will be defined as a feature vertex and be preserved if the angle of two adjacent edges is less than the feature vertex angle.

Edge Angle If Feature Recognition is on, an edge on the input mesh will be defined as a feature edge and be preserved if the angle between the normals of two adjacent triangles is greater than the feature edge angle.

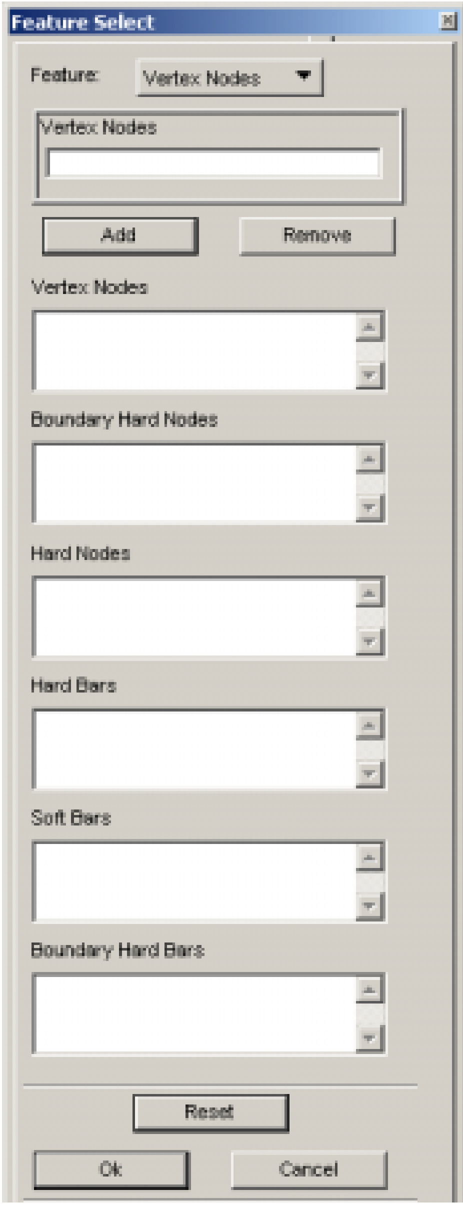
Use Selection Values If checked, all the feature entities selected on the Feature Selection form will be used as input, allowing users to pick the feature entities they want to preserve.

Feature Selection For users to select feature entities: vertex nodes, boundary seeds, hard nodes, hard bars and soft bars.

2D Element List Input tria or quad mesh.

Global Edge Length Specifies the mesh size that will be used to create the output mesh. Users can input this value or let the program calculate it for them.

Feature Select



Feature Allows you to pick the feature entities to preserve: vertex nodes, boundary seeds, hard nodes, hard bars and soft bars.

Feature Selection Box The selected entities will be added to or removed from the corresponding entity list.

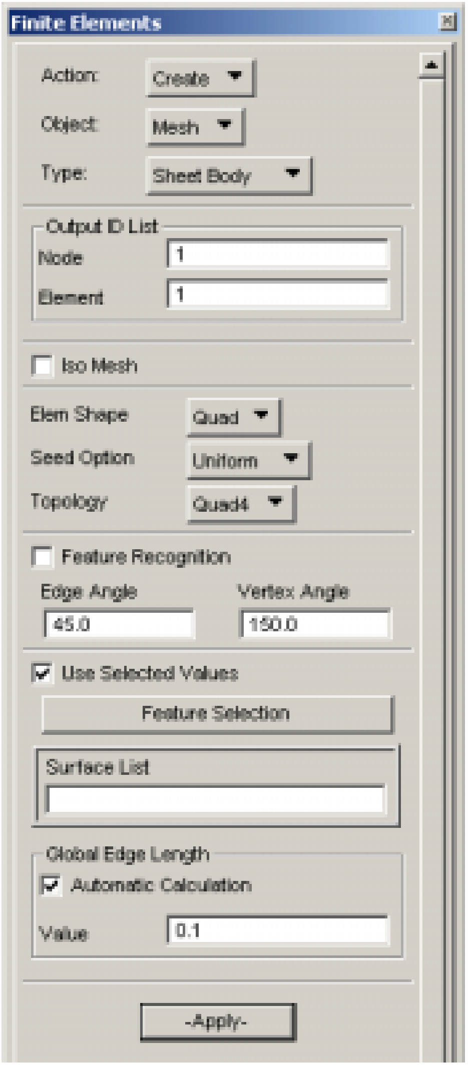
Vertex Nodes The vertex nodes are used to define 4 corner nodes on a 4-sided region when the Iso Mesh toggle is on.

Boundary Hard Nodes Select the boundary nodes you want to preserve. You have to select boundary nodes if you choose the seed option Defined Boundary.

Hard Nodes	Select the hard nodes you want to preserve. The nodes may not be on the input mesh. The program will project the nodes onto the input mesh before meshing.
Hard Bars	Select bar elems as hard feature edges on the interior of the input mesh. A hard edge, together with its end nodes, will be preserved on the output mesh. The bar element may not be on the input mesh. The program will project the nodes onto the input mesh before meshing.
Soft Bars	Select bar elems as soft feature edges on the interior of input mesh. A soft edge is a part of a feature line. The feature line will be preserved on the output mesh, but its nodes may be deleted or moved along the feature line. The bar element may not be on the input mesh. The program will project the nodes onto the input mesh before meshing.
Boundary Hard Bars	Select bar elements on boundary of the input mesh.

Sheet Body

Sheet Body Mesh operates on a sheet body, defined as a collection of congruent surfaces without branch edges. It meshes a sheet body as a region. The elements on the output mesh may cross surface boundaries. This feature is very useful in meshing a model that has many small sliver surfaces. With this mesher, users can define ridge features by selecting them or using the automatic feature recognition option. The feature curves and points on the sheet body will be preserved on the output mesh. Also, the mesher will recognize 4-sided regions and create good iso-meshes on these regions.



Iso Mesh toggle

If checked, an iso-mesh will be created on a 4-sided region. You need to select 4 corner nodes in the data box Feature Selection / Vertex Points.

Element Shape

Element Shape consists of:

- Quad
- Tria

Seed Option

- **Uniform.** The mesher will create new boundary nodes based on the input global edge length.
- **Existing Vertices.** All vertices on the boundary of the model will be preserved

Topology

- Quad4
- Tria3

Feature Recognition If checked, the feature points and curves on the model will be defined automatically based on feature edge angles and vertex angles, and be preserved on the output mesh.

Vertex Angle If Feature Recognition is on, a vertex on the model will be defined as a feature vertex and be preserved if the angle at the vertex is less than the feature vertex angle.

Edge Angle If Feature Recognition is on, an edge on the model will be defined as a feature edge and be preserved if the angle between the normals of two adjacent regions is greater than the feature edge angle.

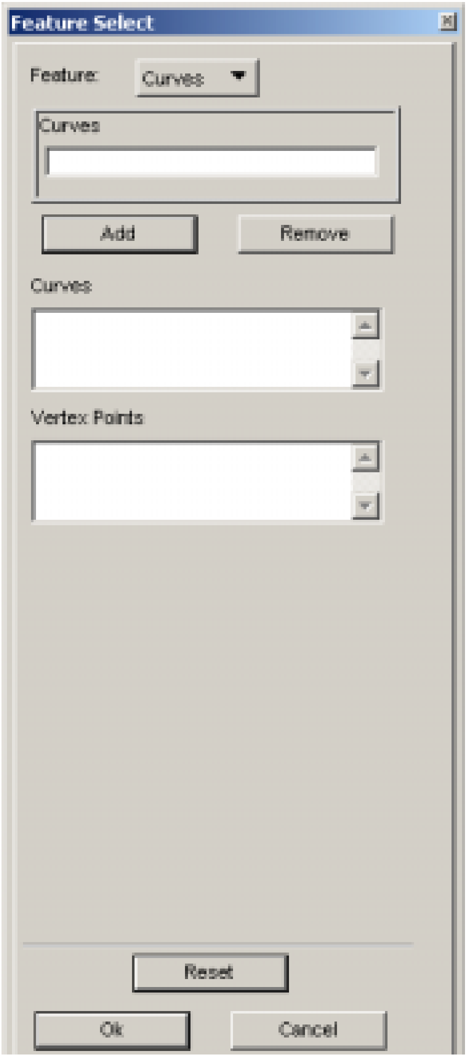
Use Selected Values If checked, all the feature entities selected on the Feature Selection form will be used as input, allowing users to pick the feature entities they want to preserve.

Feature Selection For users to select feature entities to preserve: curves and vertex points.

Surface List The input surfaces will be grouped into regions based on free or non-congruent surface boundary curves.

Global Edge Length Specifies the mesh size that will be used to create the output mesh. Users can input this value or let the program calculate it for them.

Feature Select



Feature Allows users to pick the feature entities they want to preserve: curves and vertex points.

Feature Selection Box The selected entities will be added to or removed from the corresponding entity list.

Curves Select the feature curves on the interior of the model you want to preserve.

Vertex Points Select the feature points on the boundary or the interior of the model you want to preserve.

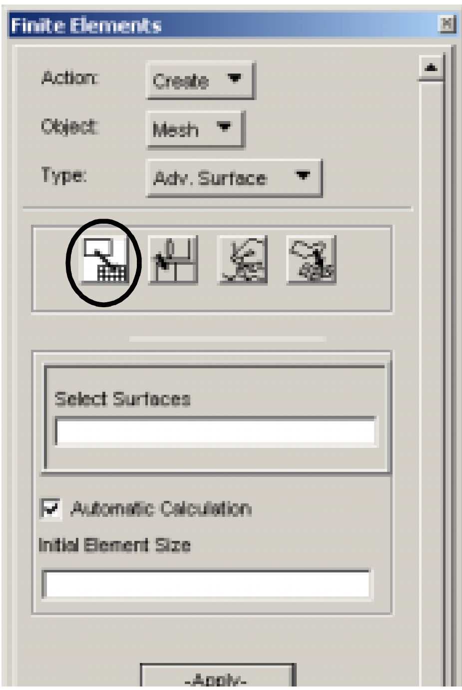
Advanced Surface Meshing

Advanced Surface Meshing (ASM) is a FEM based process that allows you to automatically create a mesh for a complex surface model. ASM can use a surface geometry that is either congruent or noncongruent. Noncongruent surfaces can contain sliver surfaces, tiny edges, and surface overlaps. For noncongruent surfaces, the geometry is first converted into pseudo surfaces represented by a mesh. Tools are provided for cleaning up the pseudo-surfaces as required. These modified pseudo surfaces can then be meshed to generate a quality quad-dominant mesh.

Application Form

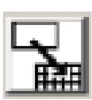
To access the ASM Application form, click the Elements Application button to bring up Finite Elements Application form, then select Create / Mesh / Adv Surface as the Action / Object / Method combination.

1. Initial Mesh Form



Process Icon Specifies the step in the ASM process.

- **1. Initial Mesh** The Initial Mesh icon is selected as the default to begin the ASM process. Converts geometry into pseudo surfaces.



Select Surface Specifies the surgace geometry to be converted into pseudo surfaces.

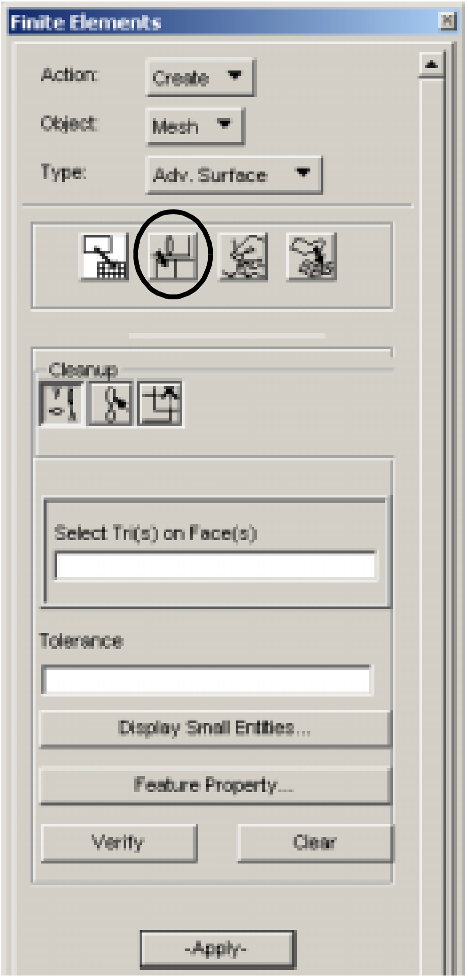
Automatic Calculation

When Automatic Calulation is turned on, the "Initial Element Size" will be set automatically. Turn the toggle off for manual entry.

Initial Element Size

The element size used to generate the pseudo surface. This size will define how well the pseudo surface will represent the real surface. A good "Initial Element Size" will be 1/5 the size of the desired final mesh size.

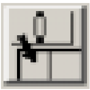
2. Cleanup



Process Icon

Specifies the step in the ASM process.

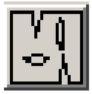
- **2. Cleanup**



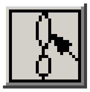
Provides tools to help in stitching gaps between the pseudo surfaces. Both automated and interactive stitching tools are available to make the model congruent.

Cleanup Tools

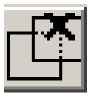
- **Auto Stitch** — Stitch all the gaps with sizes less than the specified tolerance on the selected pseudo surfaces.



- **Stitch Gap** — Stitches gaps formed by the selected free edges without checking the tolerance.

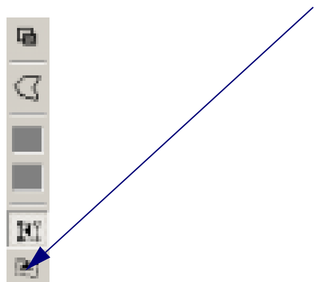


- **Delete Face** — Deletes the elements on the surface and group associated with the surface from the database. Surfaces that are “fixed” will not be deleted.



Select Tri(s) on Face(s) — Specifies pseudo surfaces by selecting the guiding tria-elements on faces. If one or more tria-elements on a surface are selected, the surface is selected.

Select Elem Free Edge — Selects free edges. To cursor select the free edges, use the “Free edge of 2D element “ icon.



Tolerance — Gaps with sizes less than this value will be stitched.

Display Small Entities — Brings up a sub-form to display short surface edges, non-manifold edges and small surfaces on the model.

Feature Property — Brings up a sub-form to set, modify and show surface feature properties, including mesh size and feature state (free or fixed) of a face.

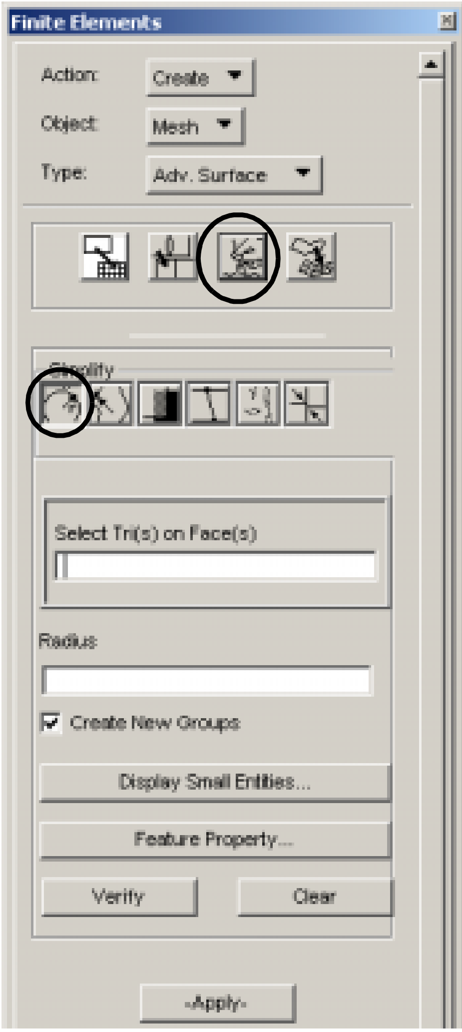
Verify — Displays the free edges on the model.

Clear — Clears the free edge display.

3. Simplify

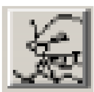
Choosing the Simplify Process Icon provides you with six tools to simplify the pseudo surfaces and prepare them for final meshing.

Tool 1. Simplify/Auto Fill



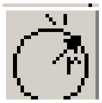
Process Icon Specifies the step in the ASM process.

- **3. Simplify** Simplifies the cleaned up model to create a desired mesh. Tools are available to remove holes, break and merge pseudo faces, and collapse edges. Surfaces can be tagged as fixed or free for the simplification process.



Simplify Tools

- **Auto Fill** — Fills the holes on the model whose radii is less than the specified hole radius.



Select Tri(s) on Face(s) — Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Radius — The specified hole radius.

Create New Groups — If this toggle is checked, then the hole will be filled by a new pseudo-surface. If the toggle is not checked and if the hole is inside one pseudo-surface, the the hole will be removed from the surface. However, if the hole is bounded by multiple pseudo-surfaces, then the hole is replaced by a new pseudo-surface.

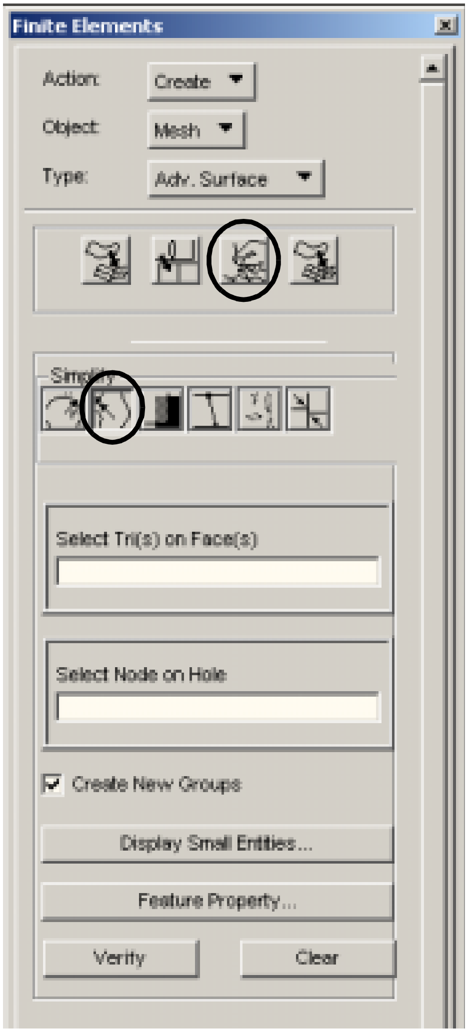
Display Small Entities — Brings up a sub-form to display short surface edges, non-manifold edges and small surfaces on the model.

Feature Property — Brings up a sub-form to set, modify and show surface feature properties, including mesh size and feature state (free or fixed) of a surface.

Verify — Displays the free edges on the model.

Clear — Clears the free edge display.

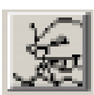
Tool 2. Simplify/Fill Hole



Process Icon

Specifies the step in the ASM process.

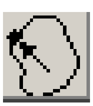
- **3. Simplify**



Simplifies the cleaned up model to create a desired mesh. Tools are available to remove holes, break and merge pseudo faces, and collapse edges. Surfaces can be tagged as fixed or free for the simplification process.

Simplify Tools

- **Fill Hole**



Fills holes identified by selecting both the pseudo-surfaces that enclose the hole and one or more nodes on the hole boundary.

Select Tri(s) on Face(s) Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Select Node on Hole Specify one or more nodes on the hole boundary.

Create New Groups If this toggle is checked, then the hole will be filled by a new pseudo-surface. If the toggle is not checked and if the hole is inside one pseudo-surface, the the hole will be removed from the surface. However, if the hole is bounded by multiple pseudo-surfaces, then the hole is replaced by a new pseudo-surface.

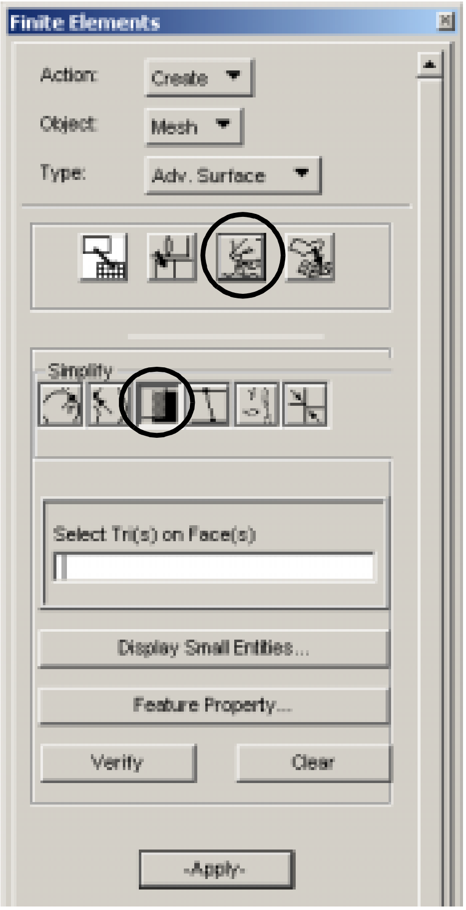
Display Small Entities Brings up a sub-form to display short surface edges, non-manifold edges and small surfaces on the model.

Feature Property Brings up a sub-form to set, modify and show surface feature properties, including mesh size and feature state (free or fixed) of a surface.

Verify Displays the free edges on the model.

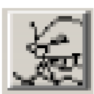
Clear Clears the free edge display.

Tool 3. Simplify/Merge Face



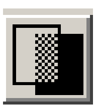
Process Icon Specifies the step in the ASM process.

- **3. Simplify** Simplifies the cleaned up model to create a desired mesh. Tools are available to remove holes, break and merge pseudo faces, and collapse edges. Surfaces can be tagged as fixed or free for the simplification process.



Simplify Tools

- **Merge Face** Merges selected pseudo surfaces into a new face. After merging, the selected pseudo surfaces will be deleted. This tool won't merge surfaces that are "fixed".



Select Tri(s) on Face(s) Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

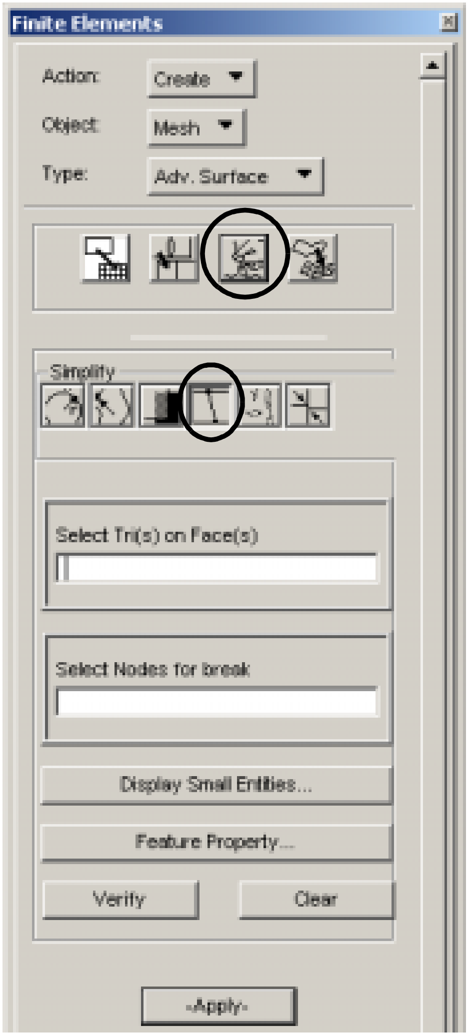
Display Small Entities Brings up a sub-form to display short edges, non-manifold edges and small surfaces on the model.

Feature Property Brings up a sub-form to set, modify and show face feature properties, including mesh size and feature state (free or fixed) of a surface.

Verify Displays the free edges on the model.

Clear Clears the free edge display.

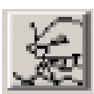
Tool 4. Simplify/Split Face



Process Icon

Specifies the step in the ASM process.

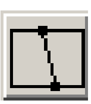
- **3. Simplify**



Simplifies the cleaned up model to create a desired mesh. Tools are available to remove holes, break and merge pseudo faces, and collapse edges. Surfaces can be tagged as fixed or free for the simplification process.

Simplify Tools

- **Split Face**



Splits a pseudo surface along the cutting line connecting two selected boundary nodes. The cutting line should divide the surface into two disconnected parts and should not intersect the face boundary at more than two points. This tool won't split surfaces that are "fixed."

Select Tri(s) on Face(s) Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Select Nodes for break Select two boundary nodes on the pseudo surface.

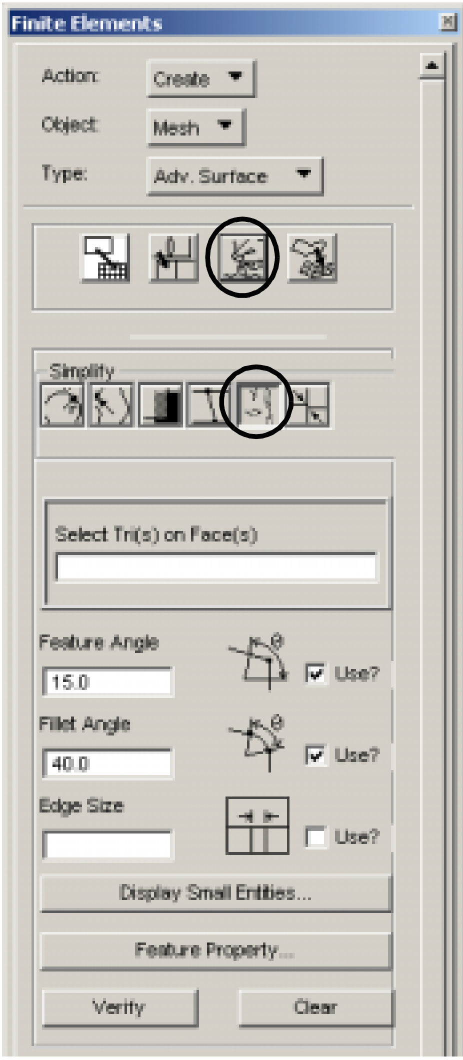
Display Small Entities Brings up a sub-form to display short edges, non-manifold edges and small surfaces on the model.

Feature Property Brings up a sub-form to set, modify and show face feature properties, including mesh size and feature state (free or fixed) of a surface.

Verify Displays the free edges on the model.

Clear Clears the free edge display.

Tool 5. Simplify/Auto Merge



Process Icon

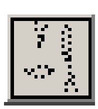
Specifies the step in the ASM process.

- **3. Simplify**

Simplifies the cleaned up model to create a desired mesh. Tools are available to remove holes, break and merge pseudo faces, and collapse edges. Surfaces can be tagged as fixed or free for the simplification process.

Simplify Tools

- **Auto Merge**



Uses several criteria (3 are exposd to you) to simplify the model. Use the check boxes to activate and deactivate these criteria. During simplification, faces with edges smaller than the Edge Size specified can be merged and surfaces with Feature Angles and Fillet Angles less than that specified can also be merged. In the merging process, additional criteria are used to maximize the creation of 3 or 4-sided surfaces and reduce the number of T-sections. Surfaces that are “Fixed” will be ignored during simplication.

Select Tri(s) on Face(s) Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Feature Angle The angle of a FEM edge is defined as the angle between the normals of its adjacent triangles. A topological edge is made of one or more FEM edges. The Feature Angle of a topological edge is the maximal angle of all the FEM edges that represent this edge.

Fillet Angle A bent angle of a cross section on a surface is the angle between the normals of the two sides of the cross section. The fillet angle of a surface is the maximal bent angle of all cross sections on the surface.

Edge Size The size of a surface is defined in different ways. The size of a circular surface is its diameter; the size of a ring region is the length of its cross section; and the size of a 3 or 4-sided and other general region is the length of the shortest side.

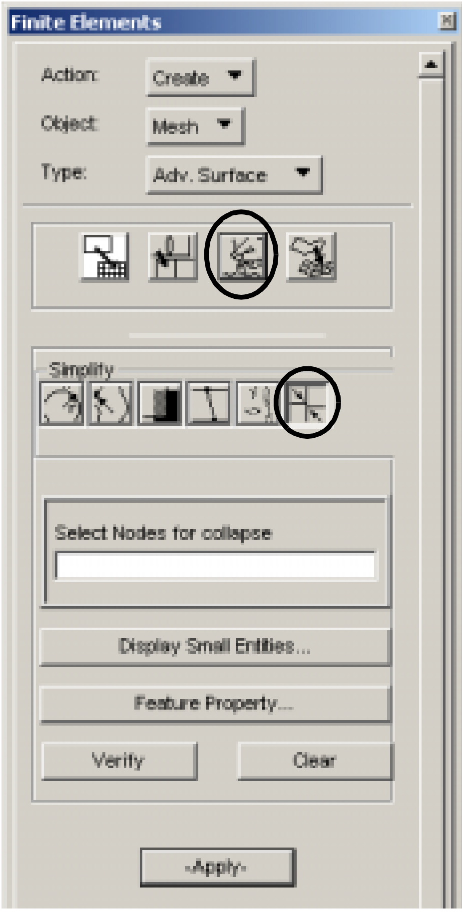
Display Small Entities Brings up a sub-form to display short edges, non-manifold edges and small surfaces on the model.

Feature Property Brings up a sub-form to set, modify and show face feature properties, including mesh size and feature state (free or fixed) of a surface.

Verify Displays the free edges on the model.

Clear Clears the free edge display.

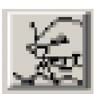
Tool 6. Simplify/Collapse Edge



Process Icon

Specifies the step in the ASM process.

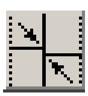
- **3. Simplify**



Simplifies the cleaned up model to create a desired mesh. Tools are available to remove holes, break and merge pseudo faces, and collapse edges. Surfaces can be tagged as fixed or free for the simplification process.

Simplify Tools

- **Collapse Edge**



Collapses a short edge on the model. Here an edge means a feature edge, not an element edge. A feature edge of a face is bounded by two vertices (corner points or branch points).

Select Nodes for collapse

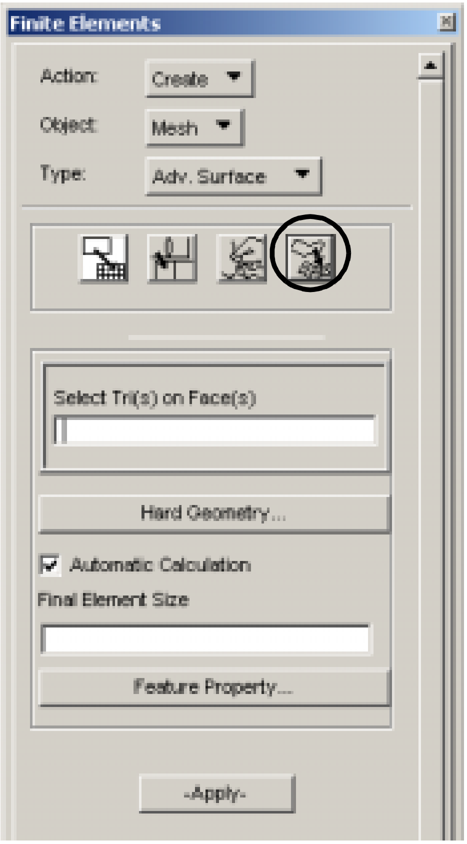
Select two vertices on an edge to be collapsed.

Display Small Entities

Brings up a sub-form to display short edges, non-manifold edges and small surfaces on the model.

Feature Property	Brings up a sub-form to set, modify and show surface feature properties, including mesh size and feature state (free or fixed) of a surface.
Verify	Displays the free edges on the model.
Clear	Clears the free edge display.

4. Final Mesh



Process Icon

Specifies the step in the ASM process.

- **4. Final Mesh**

  Meshes the simplified model to generate a quad-dominant mesh. Mesh sizes can be defined to generate the desired mesh

Select Tri(s) on Face(s)

Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Hard Geometry

Bring up a sub-form to select hard points and hard curves on the models. The hard points and hard curves will be preserved on the final mesh.

Automatic Calculation

If this toggle is checked, the program will calculate an approximate final element size once the pseudo surfaces are selected.

Final Element Size

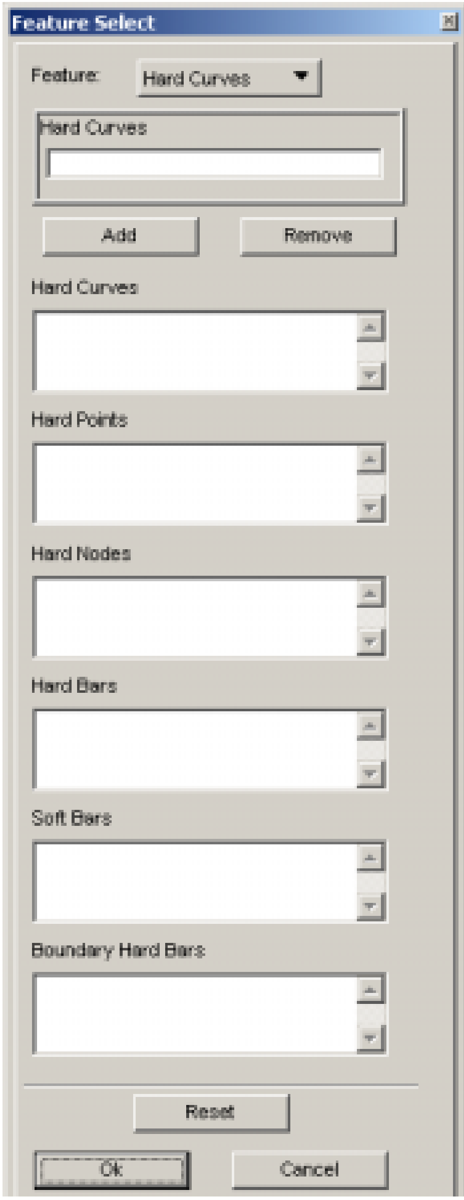
Specify the element size for the final quad mesh. The size will not override any specified mesh sizes on pseudo surfaces.

Feature Property

Brings up a sub-form to set, modify and show surface feature properties, including mesh size and feature state (free or fixed) of a surface.

Final Mesh/Hard Geometry

Defines hard curves, points, nodes, bars, soft bars and boundary hard bars. The defined hard points, hard curves, hard bars and soft bars will be preserved on the final mesh.



Hard Curves Select and deselected hard curves on the model.

Hard Nodes Select and deselected hard nodes on the model.

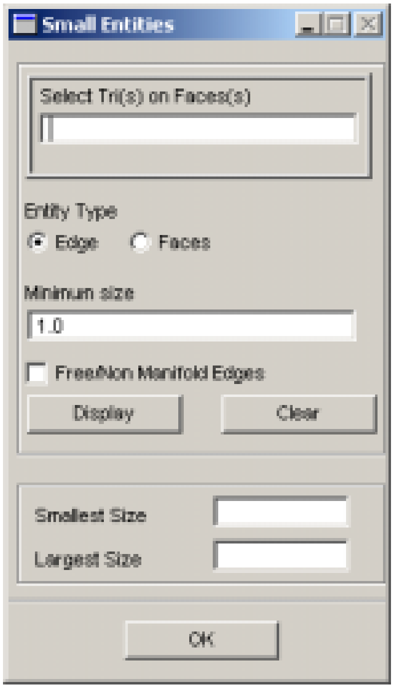
Hard Bars Select and deselected hard bars on the model. The hard bar, together with its end nodes, will be preserved on the final mesh.

Soft Bars Select and deselected soft bars on the model. A soft bar is a segment of a feature line on the model. The feature line will be preserved on the final mesh, but its nodes may be deleted or moved along the feature line.

Boundary Hard Bars	Select and deselected boundary hard bars on the model. Here boundary means the boundary of the selected mesh region.
Reset	Reset the data select box.
OK	Confirm the selection and return to the Final Mesh Form.
Cancel	Cancel the selection and return to the Final Mesh Form.
Reset	Reset the data select box
OK	Confirm the selection and return to the Final Mesh form.
Cancel	Cancel the selection and return to Final Mesh form.

Display Small Entities

The Small Entities dialog box can be used to display small edges, free edges, non-manifold and small faces in the pseudo geometry. .



Select Tri(s) on Face(s) Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Entity Type This toggle determines whether the small entities of edges or faces are to be displayed.

Minimum Size All edges smaller than this length or all surfaces smaller than this area will be identified and displayed. This size will not be used if Free / Non Manifold Edges are to be displayed.

Free/Non Manifold Edges If this toggle is checked, the free and non-manifold edges will be displayed.

Display This will display the small edges or faces, or free / non-manifold edges.

Clear This will clear the display.

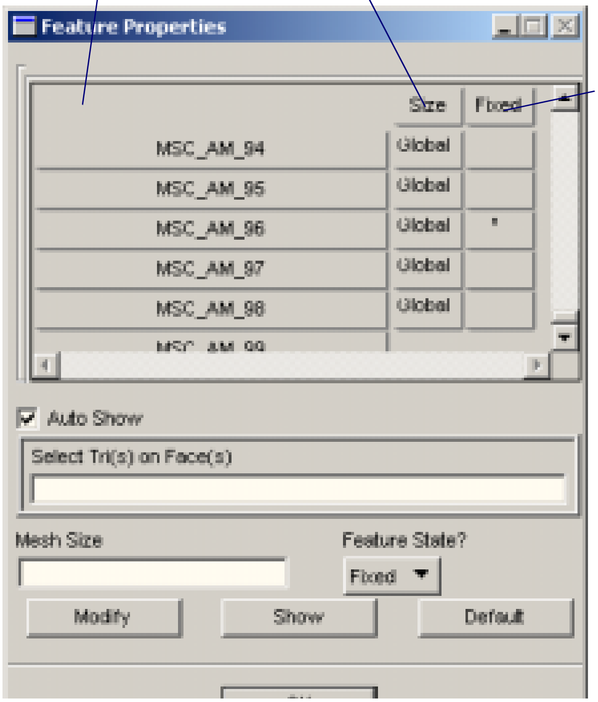
Smallest/Largest Size The smallest and the largest edge or face area will be calculated and displayed here.

Feature Properties

The “mesh size” and the “feature state” properties of pseudo surfaces can be set, modified, and displayed using this form..

The group name that identifies the pseudo surface

Element size associated with the pseudo surface



The state of a pseudo surface can be “Free” or “Fixed”. Unlike a fixed pseudo surface a free pseudo face can be modified using “Cleanup” and “Simplify” operations. An asterisk will appear in this cell if the surface is fixed. In this example, the surface identified by “MSC_AM_96” is fixed whereas “MSC_AM_97” is free to be modified.

Auto Show If checked, the Mesh Size and Feature State of the selected pseudo surfaces will be displayed in the Command History window.

Select Tri(s) on Face(s) Specify pseudo surfaces by selecting the guiding tria-elements on the surfaces. A surface is selected if at least one of its tri elements is picked.

Mesh Size Mesh size for the pseudo surfaces.

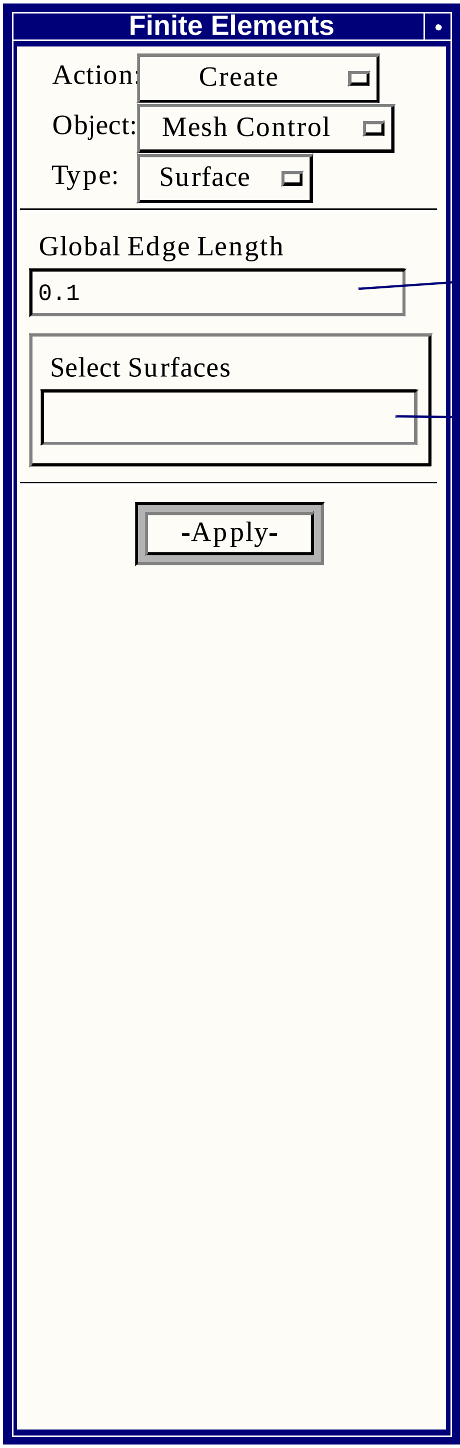
Feature State Toggle to change the Feature State of selected pseudo surfaces.

Modify Modify the mesh sizes or feature states of pseudo surfaces selected.

Show Shows the mesh sizes and feature states of the selected pseudo surfaces in the Command History window.

Default Sets the default values for the mesh sizes (Global value) and the feature states (Free) for the selected pseudo surfaces.

2.4 Mesh Control



User defined edge length for surfaces selected below.

List of surfaces for which the mesh control should be applied to.

Auto Hard Points Form

Using the Create / Mesh Control / Auto Hard Points form creates hard points on a specified set of surfaces automatically. This program creates hard points at two kinds of points on surface boundaries: T Points - A T point is defined as an interior point of a surface edge which is close to a vertex or an existing hard point on an edge in another surface within the t-point tolerance. The t-point tolerance is equal to one twentieth of the target element edge length. Placing a hard point at a T-point will help meshers create a congruent mesh on a noncongruent model. There is a noncongruent model in **Figure 2-14**. The auto hard point creation program creates a T-point at the T-junction of three surfaces and marks it by a small triangle. The new hard point forces the mesher to place a node at the T-junction when meshing surface 1 and the mesh created is a congruent mesh (**Figure 2-15**). **Figure 2-16** shows the mesh without hard point creation.

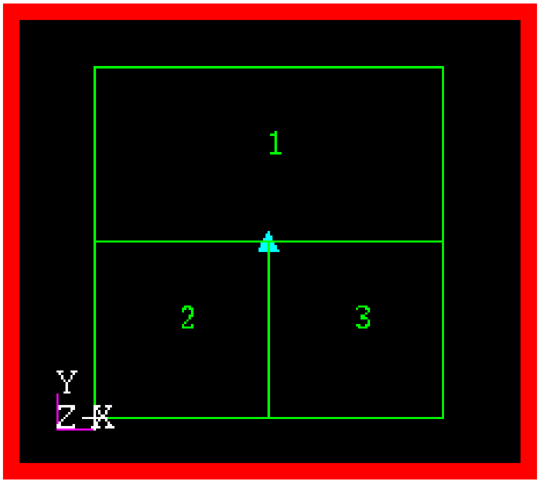


Figure 2-14 T-Point Creation

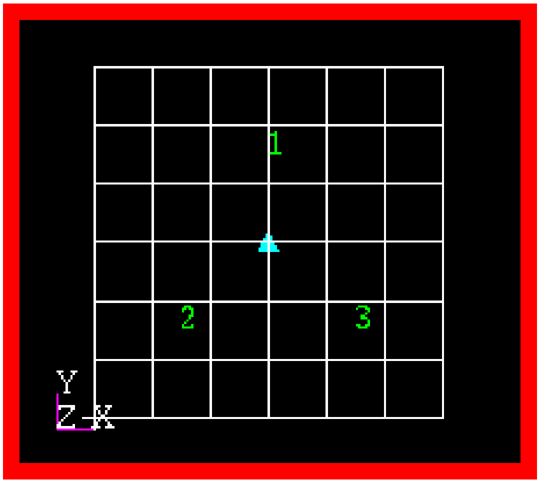


Figure 2-15 Mesh with Hard Point Creation

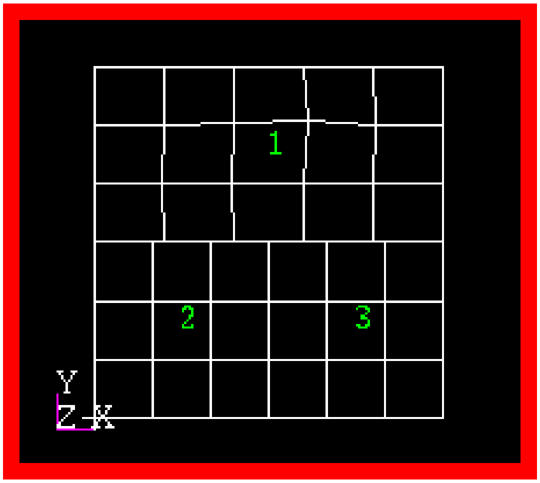


Figure 2-16 Mesh Without Hard Point Creation

Neck Points. A neck point is defined as an end point of a short cross section on a surface. A cross section on a surface is short if its length is less than the neck-point tolerance.The neck-point tolerance is equal to 1.5 times the target element edge length. Placing a hard point at a neck point will help meshers create better meshes on narrow surfaces. Neck points can be created recursively by neck-point propagation. In Figure 2-17, the two neck-points on the boundary of surface 1 were created first and the remain four neck points were created by neck point propagation from one small surface to another until the path reached the outer boundary of the model. The new hard points will help mesher line up the boundary nodes and create a good mesh on the narrow surfaces (Figure 2-18). Figure 2-19 shows the mesh without hard point creation.

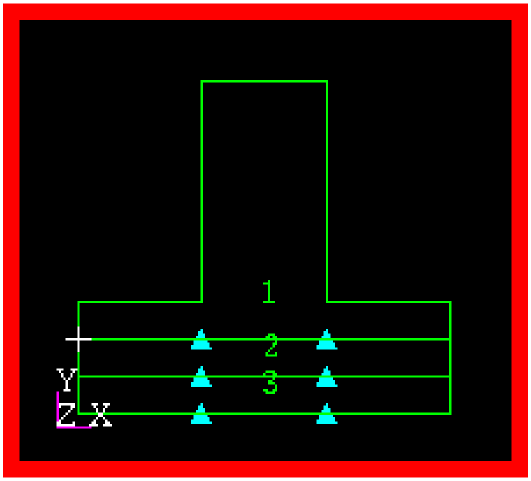


Figure 2-17 Neck Point Propagation

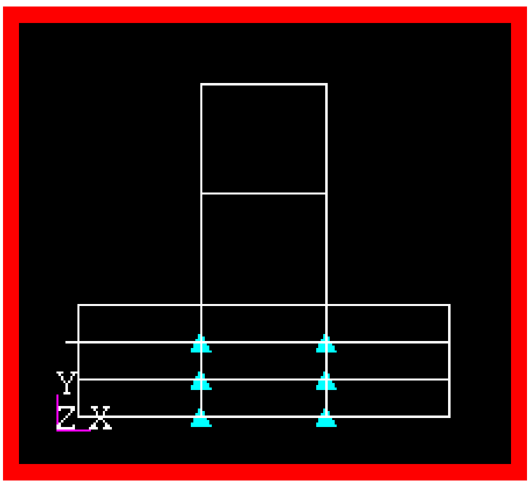


Figure 2-18 Mesh with Hard Point Creation

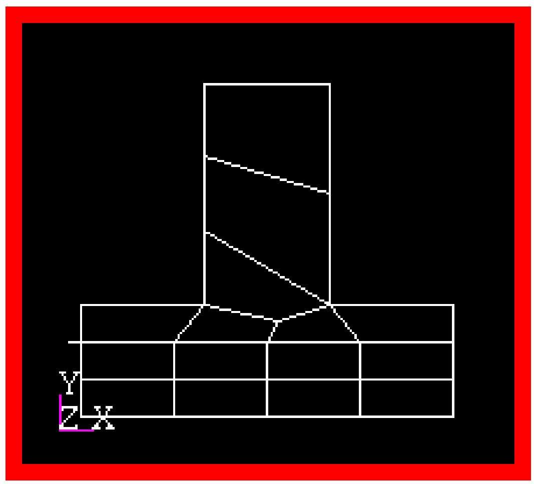
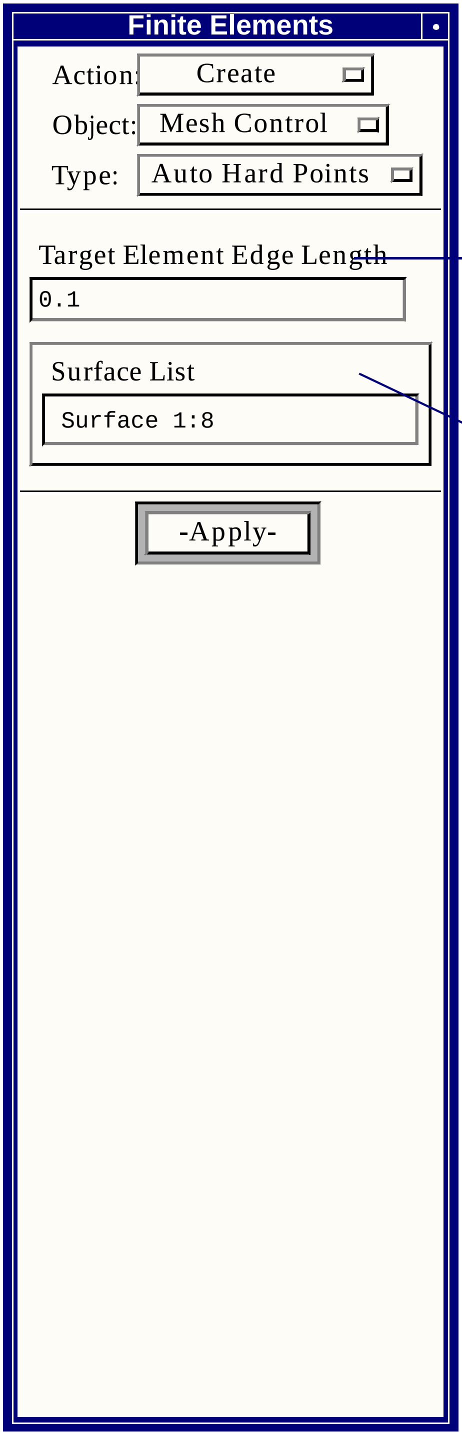


Figure 2-19 Mesh Without Hard Point Creation

Figure 2-20



Specify the target element edge length which you will use as a global edge length when you mesh the model. The target element edge length is used to compute the t-point tolerance and neck-point tolerance.

Specify a set of surfaces on which to create hard points, either by cursor selecting them or by entering the IDs from the keyboard. (Example: Surface 1:10).

CHAPTER

3

The Create Action (FEM Entities)

- Introduction
- Creating Nodes
- Creating Elements
- Creating MPCs
- Creating Superelements
- Creating DOF List
- Creating Connectors

3.1 Introduction

The following sections describe how to create individual nodes, elements, and Multi-Point Constraints (MPCs). To create a mesh of nodes and element, see **Creating a Mesh** (p. 29).

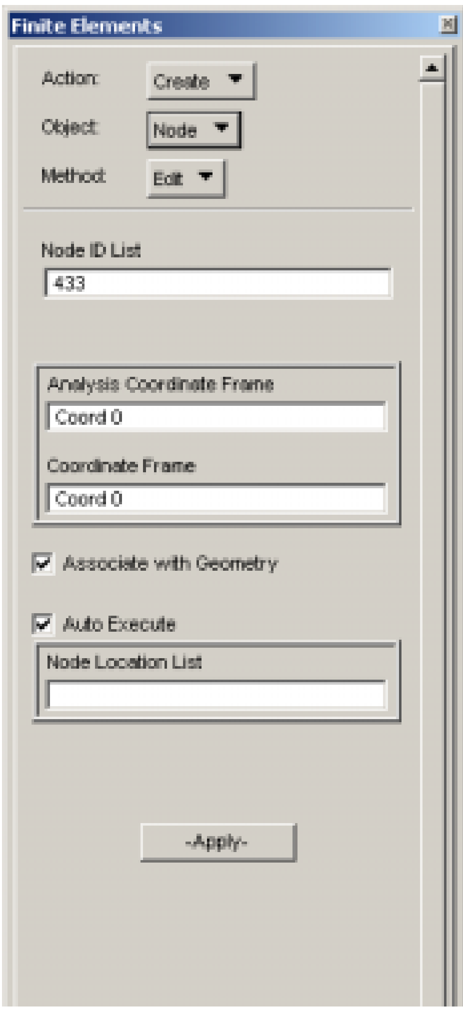
3.2 Creating Nodes

Create Node Edit

Description

The XYZ method creates nodes from defined cartesian coordinates or at an existing node, vertex or other point location as provided in the Point select menu.

Application Form



Node ID List Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame

Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame See **The Show Action** (Ch. 12).

Associate with Geometry

Indicates whether nodes should be associated with the geometry on which they are created. When the toggle is ON, nodes are associated with the point, curve, surface or solid on which they are created. Normally nodes should be associated, since loads and BCs applied to the geometry are only applicable to nodes and elements associated with that geometry. However, when selected OFF, additional methods of entering nodal location are available.

Node Location List

Specifies node locations by entering coordinates, or by using the select menu. All locations are in the Reference Coordinate Frame.

Create Node ArcCenter

Description

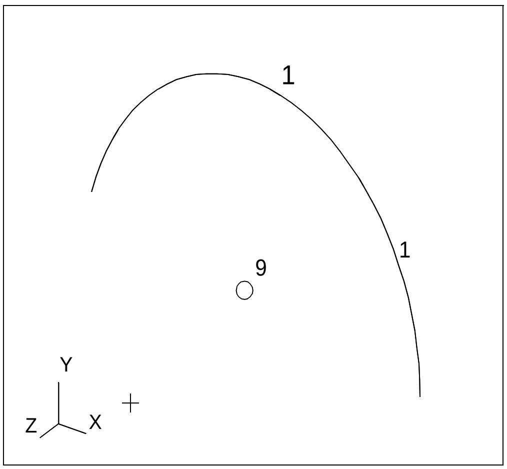
The ArcCenter method creates a node at the center of curvature of the specified curves which have a non-zero center/radius of curvature.

Examples

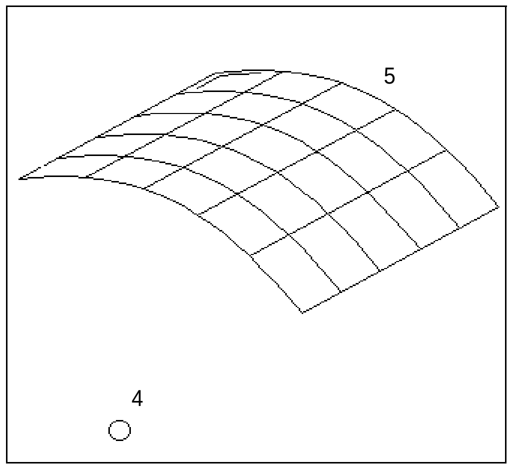
Note that, to enhance visual clarity, the display size of the points in the examples has been increased with the Display>Geometry command.

1. Node **9** was created at the center point of an arc (Curve 1).
2. Node **4** was created at the curvature center of edge 2 of surface 5 (Surface 5.2).

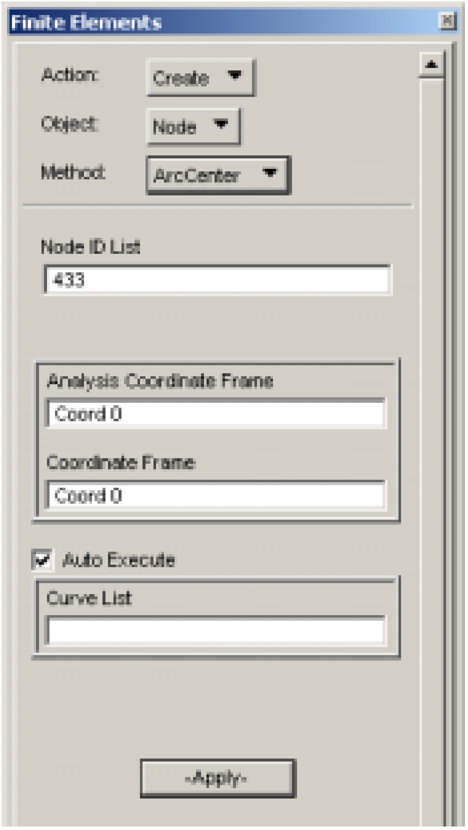
Example 1



Example 2



Application Form



Node ID List — Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame — Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame — Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame.

Curve List — Specify the existing curves or edges either by cursor selecting them or by entering the IDs from the keyboard. **Example:** Curve 1 Surface 5.1 Solid 5.1.1. The Curve Select menu that appears can be used to define how you want to cursor select the appropriate curves or edges.

Extracting Nodes

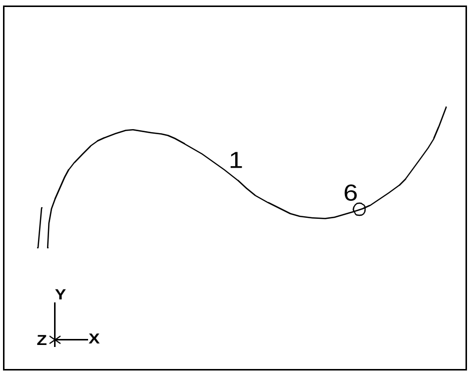
Description

With this command you can extract and display a node at any point of a curve or edge, and one or more nodes at any point of a face, at specified parametric distances from the parametric origin.

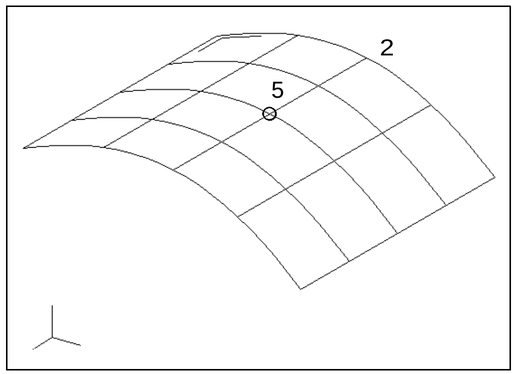
Examples

1. Node **6** of Curve **1** was extracted at a parametric distance of u=0.67.
2. Node **5** of Surface **2** was extracted at the center of the surface (u = v = 0.5).
3. Several nodes of Surface **20** were extracted within specified parametric boundaries.

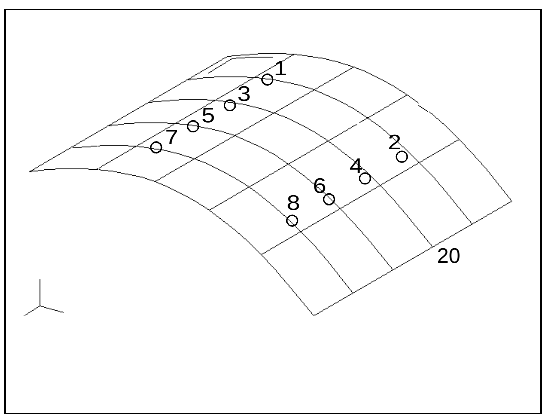
Example 1



Example 2

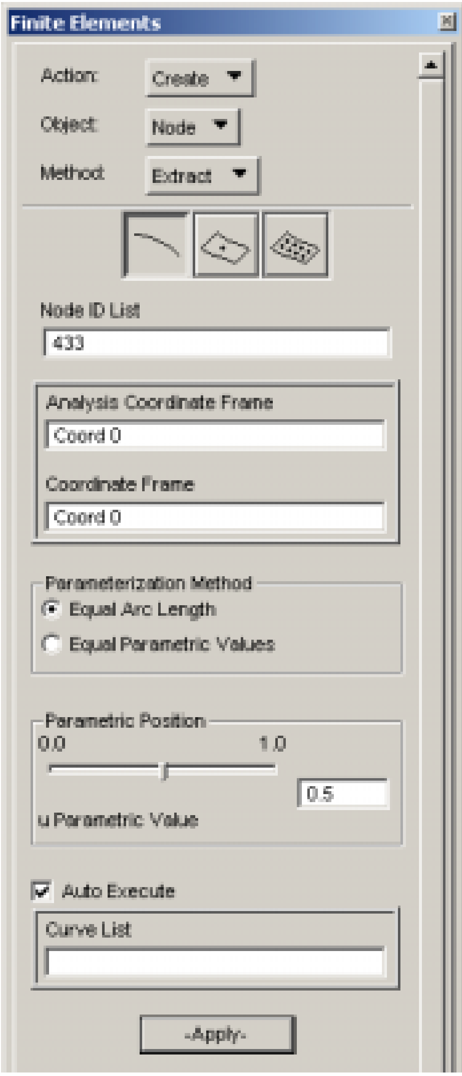


Example 3

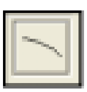


Application Form

1. Extract a node from a curve or edge



Curve Symbol



Indicates that the geometry from which a node will be extracted is a *curve*.

Node ID List

Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame

Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame

Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame.

Parameterization Method

- Equal Arc Length: The parametric distance value specified for *u* is calculated in terms of equal-length arc segments along the curve.
- Equal Parametric Values: The parametric distance value specified for *u* is calculated in terms of equal parametric values.

Parametric Position

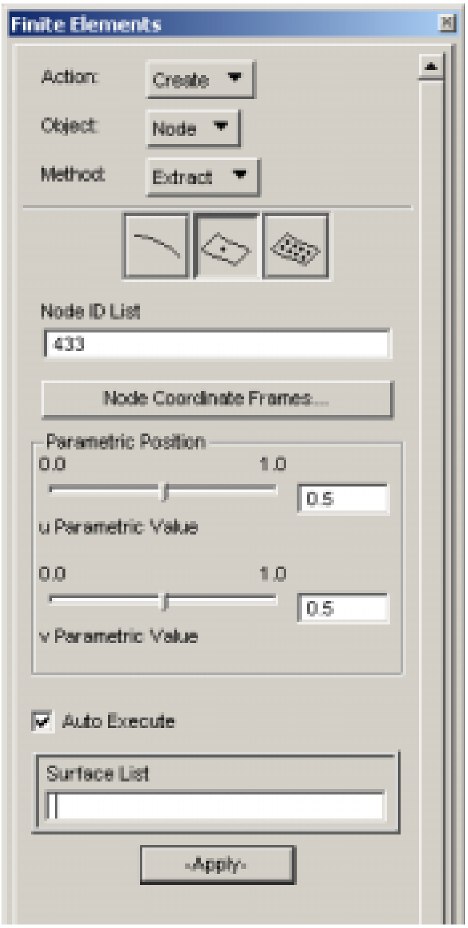
Specify the curve's or edge's $\xi_1(u)$ coordinate value, where ξ_1 has a range of $\le \xi_1 \le 1$, either by using the slide bar or by entering the value in the databox. The direction of ξ_1 is defined by the connectivity of the curve or edge. You can plot the ξ_1 direction by choosing the Parametric Direction toggle on the Geometric Properties form under the menus Display / Display Properties / Geometric.

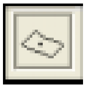
- Slide Bar: Move the slider to a desired parameter value ($0 \le u \le 1$). The databox will show numerically the position of the slider along the parametric length.
- Counter (databox): If greater accuracy is desired, type a specific value into the databox. In turn, the position of the slider will change to reflect the numerical value in the counter.

Curve List

Specify the existing curves or edges either by cursor selecting them or by entering the IDs from the keyboard. **Example:** Curve 1 Surface 5.1 Solid 5.1.1. The Curve Select menu that appears can be used to define how you want to cursor select the appropriate curves or edges.

2. Extract a node from a surface or face



	**Surface Symbol (One point)**	Indicates that: a.) the geometry from which to extract is a *surface*, and b.) only *one* node is being extracted.

Node ID List — Displays the ID of the *next* node that will be created.

Node Coordinate Frame — Select the Analysis Coordinate Frame and the Reference Coordinate Frame.

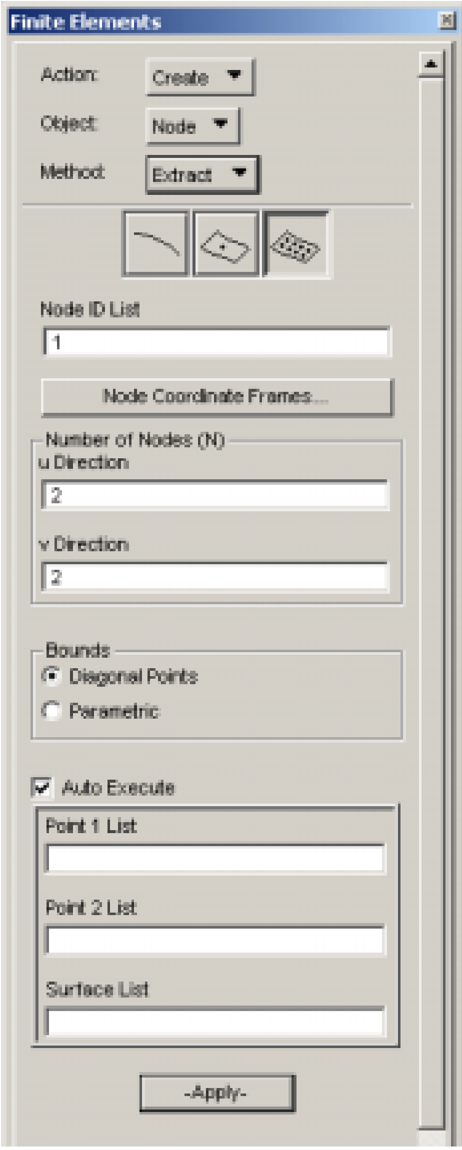
Parametric Position

- u Parametric Value — Move the slider or enter a value in the databox to specify the parametric distance from the parametric origin of the point in the *u* direction.

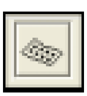
- v Parametric Value — Move the slider or enter a value in the databox to specify the parametric distance from the parametric origin of the point in the *v* direction.

Surface List — Specify the existing surfaces or faces to create nodes on, either by cursor selecting the surfaces or faces or by entering the IDs from the keyboard. **Example:** Surface 1 or Solid 5.1 The Surface Select menu that appears can be used to define how you want to cursor select the appropriate surfaces or faces.

3. Extract multiple nodes from surfaces or faces



Surface Symbol (More than one point)



Indicates that:
a.) the geometry from which to extract is a *surface*, or *face* and
b.) *several* nodes are being extracted.

Node ID List Displays the ID of the *next* node that will be created.

Node Coordinate Frame Select the Analysis Coordinate Frame and the Reference Coordinate Frame.

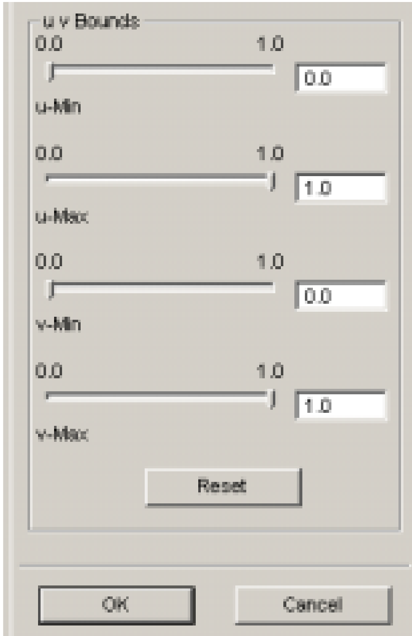
Number of Nodes

- u direction — Designates the number of nodes extracted in the *u* parametric direction.

- v direction — Designates the number of nodes extracted in the *v* parametric direction.

Bounds
Specify the Bounds as Diagonal Points when two point locations are to be used to define the boundary for the nodes to be extracted from the surface.

Parametric Bounds...
Brings up a secondary form in which you can define the upper and lower *u* and *v* limits of the area boundaries.



u-Min/u-Max
Displays the u-directional parameter values that define where the delimited area of the surface begins and ends.

v-Min/v-Max
Displays the v-directional parameter values that define where the delimited area of the surface begins and ends.

Point List 1
Point List 2
Specify the two points to define the diagonal for the points, either by cursor selecting the points or by entering the IDs from the keyboard. Example: Point 1 or Curve 1.1, Surface 1.1.1. The Point Select menu that appears can be used to define how you want to cursor select the appropriate points.

Surface List
Specify the existing surface or face to create nodes on, either by cursor selecting the surface or face by entering the IDs from the keyboard. **Example:** Surface 1 or Solid 5.1 The Surface Select menu that appears can be used to define how you want to cursor select the appropriate surface or face.

Interpolating Nodes

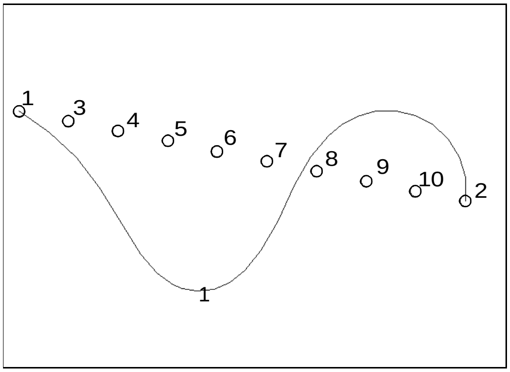
Description

This command enables you to interpolate any number of nodes between two existing points, vertices, or nodes, or two arbitrary locations cursor-selected on the screen. The interpolation between two vertices of a curve or edge may be specified either along the actual distance between the two vertices or along the curve or edge itself. Points may be either uniformly or non-uniformly distributed.

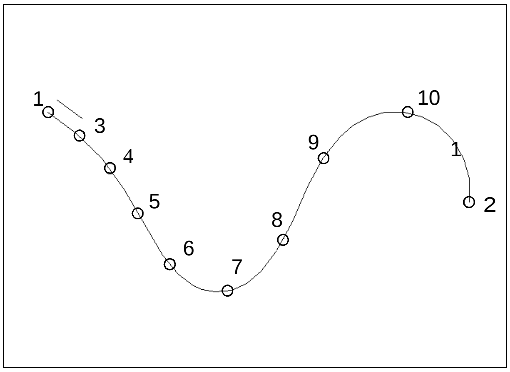
Examples

1. Interpolate eight nodes over the distance between vertices 1 and 2 of Curve 1. Points are to be equidistant.
2. Interpolate eight nodes along the curve between vertices 1 and 2 of Curve 1. Points are not uniformly spaced; the ratio of the longest to the shortest distance between two points is 3.

Example 1

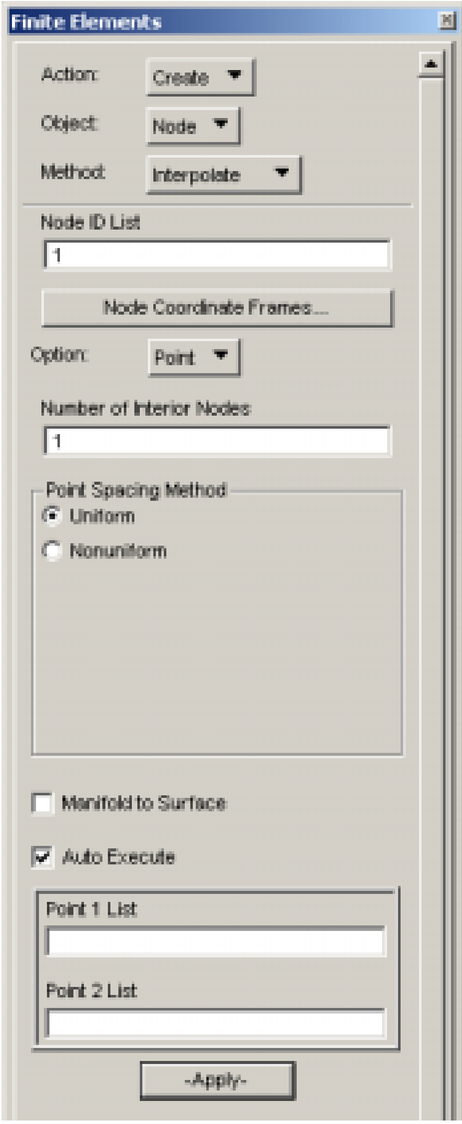


Example 2



Application Form

1. Interpolate along the distance between two points



Node ID List Displays the ID of the *next* node that will be created.

Node Coordinate Frame Select the Analysis Coordinate Frame and the Reference Coordinate Frame.

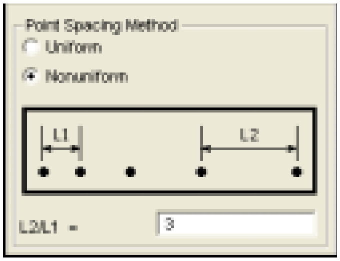
Option Choose Point or Curve

Number of Interior Nodes Enter the number of interior nodes you want to create between the specified point locations in the Point 1 and Point 2 Coordinates List.

Point Spacing Method Select either button for Uniform or Nonuniform nodes spacing for the new interior points. If Nonuniform is ON, then enter the value for L2/L1, where L2/L1 is $0 \leq L2/L1 \leq 1.0$ or $L2/L1 \leq 1.0$.

- Uniform — Interpolated nodes will be equidistant from the original nodes and from each other.

- Nonuniform — Interpolated nodes will not be spaced uniformly. The Application Form will include additional items:

  Where:

  L1 = shortest distance

  L2 = longest distance between two nodes

  When using the Curve option, L1 will be the distance between the first selected node and the first interpolated node.

Manifold to Surface

If this toggle is ON, the interpolated nodes will be projected onto a selected surface.

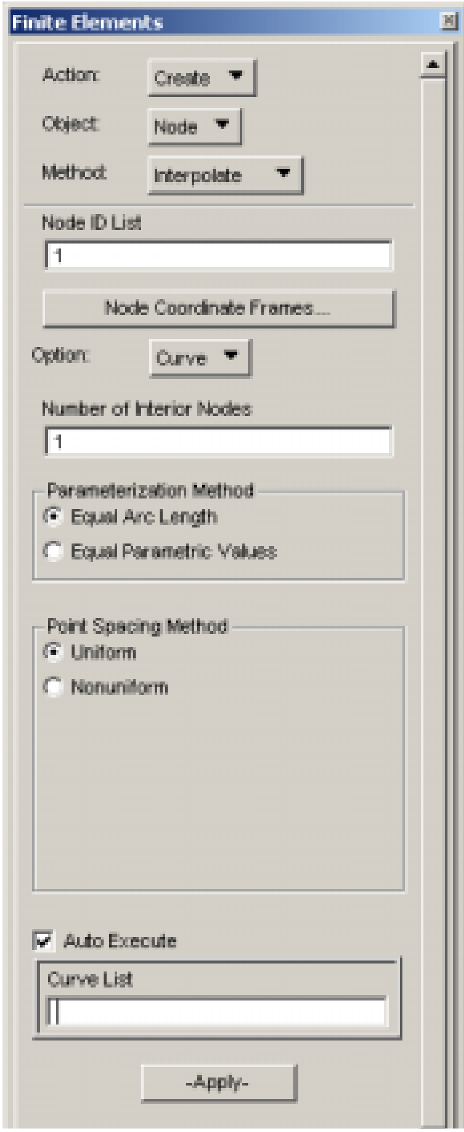
Point 1 List

Point 2 List

Specify in the **Point 1 Coordinates** listbox, the starting point location to begin the interpolation. Specify in the **Point 2 Coordinates** listbox, the ending point location to end the interpolation.

You can express the point location either by entering the location's cartesian coordinates from the keyboard, or by using the Point Select menu to cursor select the appropriate points, vertices, nodes or other point locations. **Examples:** [10 0 0], Surface 10.1.1, Node 20, Solid 10.4.3.1.

2. Interpolate Between Two Vertices Along a Curve or Edge



Node ID List Displays the ID of the *next* node that will be created.

Node Coordinate Frame Select the Analysis Coordinate Frame and the Reference Coordinate Frame.

Option Choose Point or Curve

Number of Interior Nodes Enter the number of interior nodes you want to create between the specified point locations in the Point 1 and Point 2 Coordinates List.

Parameterization Method If **Equal Arc Length** is ON, MSC.Patran will create the node(s) based on the arc length parameterization of the curve. If **Equal Parametric Values** is ON, MSC.Patran will create the point(s) based on the equal parametric values of the curve.

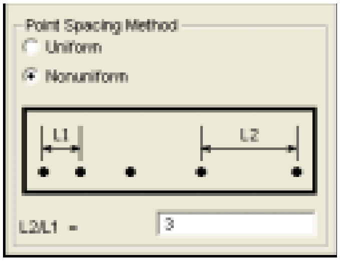
- Equal Arc Length — Parametric dimensions are calculated in terms of the length of equal arc segments along the curve.

  This method is especially useful when a number of nodes are interpolated and uniform spacing is required, because it ensures that nodes will be placed at equal arc lengths.

- Equal Parametric Values — Parametric dimensions are calculated in terms of equal parametric values.

Point Spacing Method Choose either button for Uniform or Nonuniform point spacing for the new interior point

- Uniform
- Nonuniform — Interpolated nodes will not be spaced uniformly. The Application Form will include additional items:



  Where:

  L1 = shortest distance

  L2 = longest distance between two nodes

  When using the Curve option, L1 measures the distance between the parametric origin and the first interpolated node. For determining where the parametric origin is, turn on the parametric axis (**Display>Geometry>Show Parametric Direction**).

Curve List Specify the existing curves or edges to create nodes on, either by cursor selecting the curves or edges or by entering the IDs from the keyboard. **Example:** Curve 1 Surface 5.1 Solid 5.1.1. The Curve Select menu that appears can be used to define how you want to cursor select the appropriate curves or edges.

Intersecting Two Entities to Create Nodes

Description

With this command you can create nodes at the intersections of various entity pairs and at the intersection of three planes. One intersection node will be created at each location where a point of one entity is within the global model tolerance of a point of the other.

Intersecting Entities

The following diagrams show the possible entity pairs for which intersection nodes can be created:

Curve	**Curve**
	Surface
	Plane
	Vector

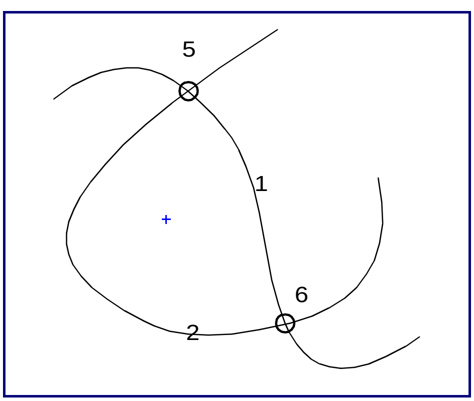
Vector	**Curve**
	Surface
	Plane
	Vector

With the exception of the curve/surface combination, “intersection” nodes may be generated even if two entities do not actually intersect. MSC.Patran will calculate the shortest distance between non-intersecting entities and place a node on each entity at the location where the shortest distance occurs.

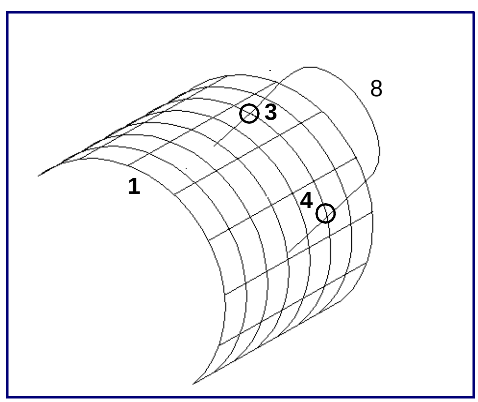
Examples

1. Nodes 5 and 6 were created at the intersections of Curve 1 and Curve 2.
2. Nodes 3 and 4 were created at the intersections of Curve 8 and Surface 1.
3. Nodes 6 and 7 were created where vector 1 would intersect Surface 5 if it were extended.
4. Nodes 8 and 9 were created at the points where the distance between Curve 3 and Curve 4 is the shortest.

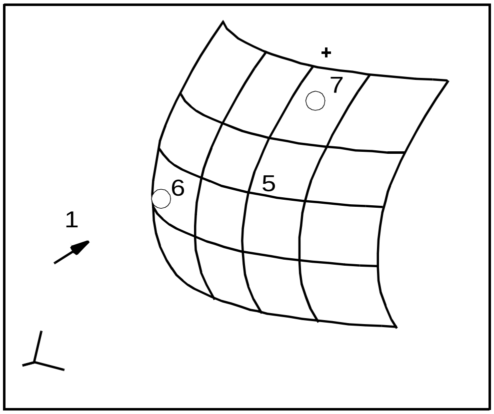
Example 1



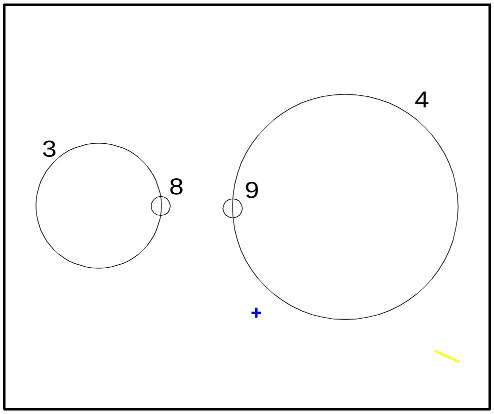
Example 2



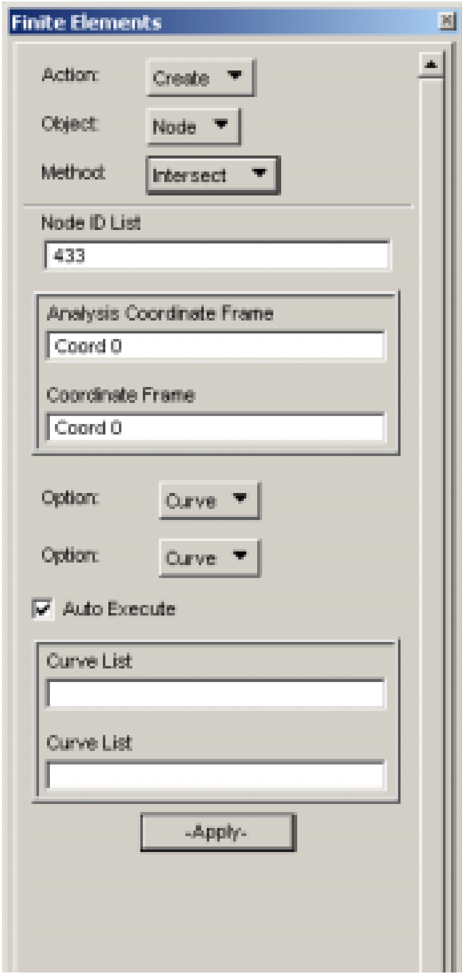
Example 3



Example 4



Application Form



Node ID List — Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame — Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame — Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame See **The Show Action** (Ch. 12).

Option (1) — Specifies a curve or vector as the first intersecting entity.

Also provides the *3 Plane* option that will create a node at the intersection of three existing plane entities.

Option (2)

Specifies a curve (edge), surface (face), plane, or vector, that is intersected by the first entity.

Listboxes

The title and contents of the listboxes will depend on what you selected for the two above options; e.g., Curve List (pick a curve or edge), Vector List (pick a vector) and others.

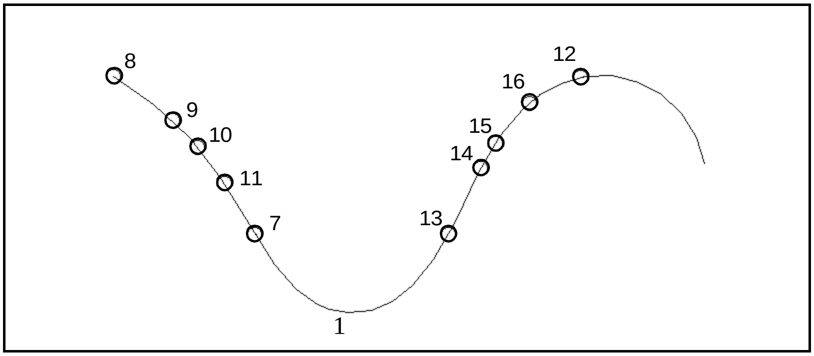
Creating Nodes by Offsetting a Specified Distance

Description

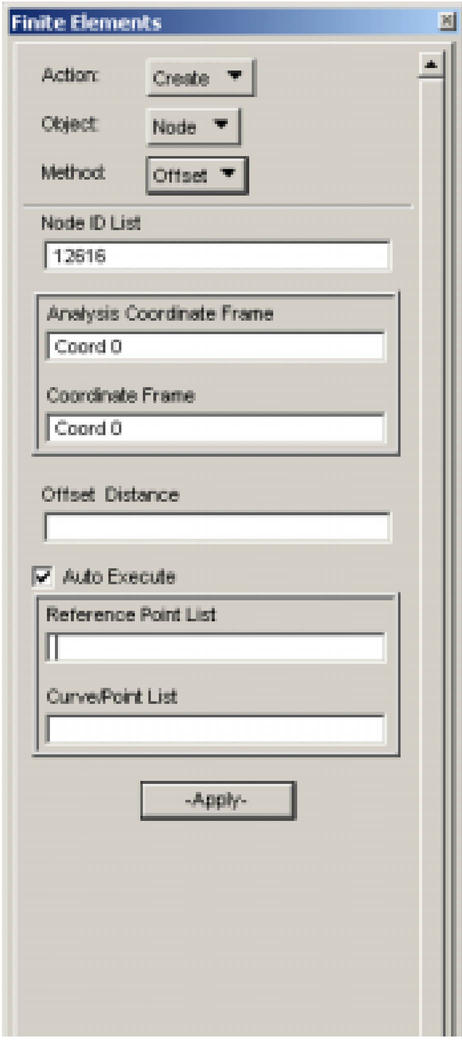
The offset method creates new nodes by offsetting existing points by a given distance along a curve or an edge.Offset distance is specified in model dimensions (not parametric!).

Example

Points 7 through 11 were offset by a distance of 8 units along Curve 1 to create nodes 12 through 14 (notice that point order is maintained).



Application Form



Node ID List — Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame — Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame — Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame See **The Show Action** (Ch. 12).

Offset Distance — Defines the distance between an offset point and its reference point.

Reference Point List

Shows the IDs of the points selected for offset. You can enter point IDs individually or in a series, or pick points on the screen. Use the Select Menu icons to “filter” your selection, e.g., for picking a vertex or intersection point.

Curve/Point List

Curve: Identifies the curve on which the points are offset.

Point: Selects a point that sets the direction of the offset.

Pick the curve, then use the Select Menu icons to focus on a particular point type. Alternately, you can double click the curve on the side to which you want to offset the point(s). The first click identifies the curve and, because the Select Menu defaults to “Any point”, with the second click you pick any point as long as it determines the correct offset direction.

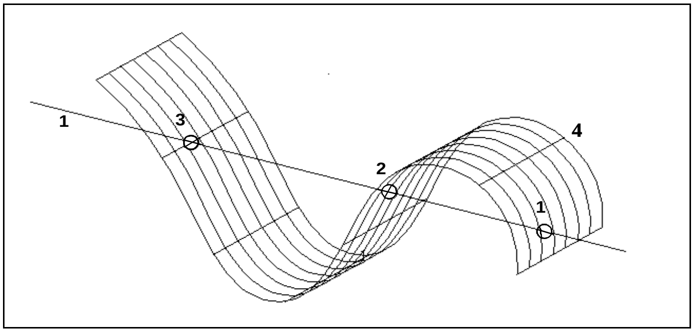
Piercing Curves Through Surfaces to Create Nodes

Description

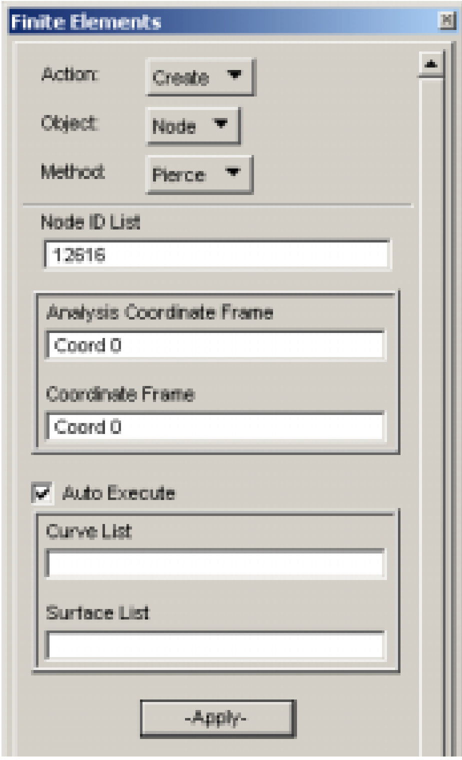
With this command you can create one or more nodes at locations where a curve or edge pierces (intersects) a surface or a face. The pierce point will be created only if there is *actual* intersection between the curve and the surface (no projected points).

Example

Created Nodes 1, 2, 3 where Curve 1 pierces Surfaces 4.



Application Form



Node ID List — Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame — Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame — Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame.

Offset Distance — Input the Model Space offset distance from an existing point on a curve (curve to be input).

Curve List — Displays the ID of the curve (or edge) that pierces the surface.

Surface List — Displays the ID of the surface that is pierced.

Projecting Nodes Onto Surfaces or Faces

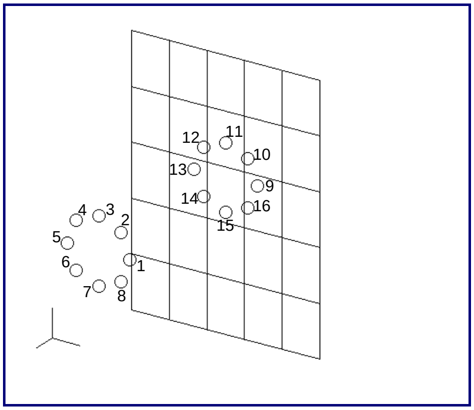
Description

Using this command you can project one or more point locations onto a curve, edge, surface, or face. The reference location that is projected may be a point entity, a specific location on other entities e.g., vertex or intersection point, a node, or a location on the screen defined by explicit coordinates or picked with the cursor. The direction of projection may be along the normal of the selected curve or surface, along any defined vector, or along the direction of the view angle of the active viewport. The original reference points may be retained or, optionally, deleted.

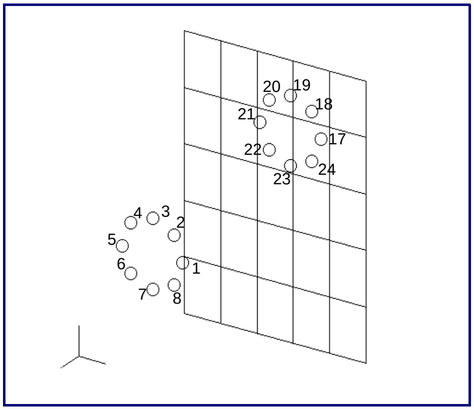
Examples

1. Nodes 1-8 are in the global XY plane. Surface 1, parallel to the XY plane, is located at Z= -3. Points 9-16 were obtained by projecting the original reference points to Surface 1 along the surface normal.
2. Nodes 17-24 are the projections of Points 1-8 along a vector V=<0.25 0.75 -3>.

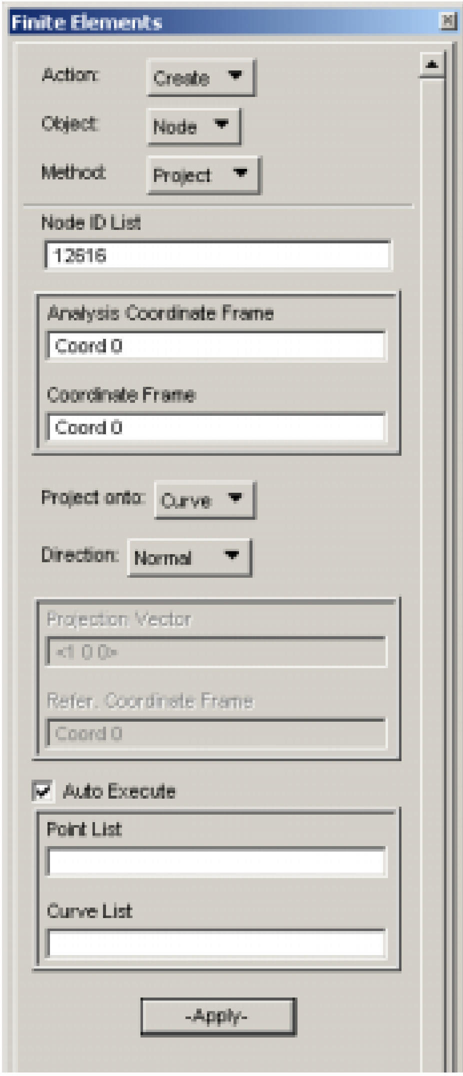
Example 1



Example 2



Application Form



Node ID List — Displays the ID of the *next* node that will be created.

Analysis Coordinate Frame — Specifies local coordinate frame ID for analysis results. The default ID is the active coordinate frame.

Coordinate Frame — Allows definition of nodal location in a local coordinate frame. Any location(s) specified in the Node Location List Select databox (on this form) are defined to be in this Reference Coordinate Frame. The default is the active coordinate frame. The Show Action will optionally report nodal locations in the Reference Coordinate Frame See The Show Action (Ch. 12).

Project onto

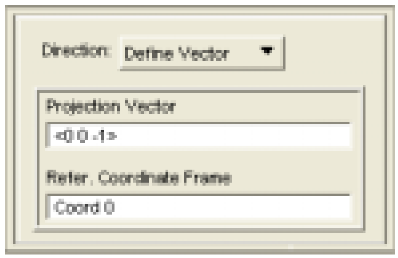
The target where projected nodes will be placed. Your options are:

- Curve
- Surface
- Plane

Direction

- **Normal** — The nodes are projected along the normal of the curve (edge) or the surface (face).
- Define Vector — The nodes are projected along an arbitrary projection vector that you define.

  The following portion of the form will become selectable:.



  Here you specify the projection vector and name the reference coordinate frame in which the vector is defined.

- View Vector — The nodes are projected along a vector whose direction is determined by the viewing angle of the current active viewport.

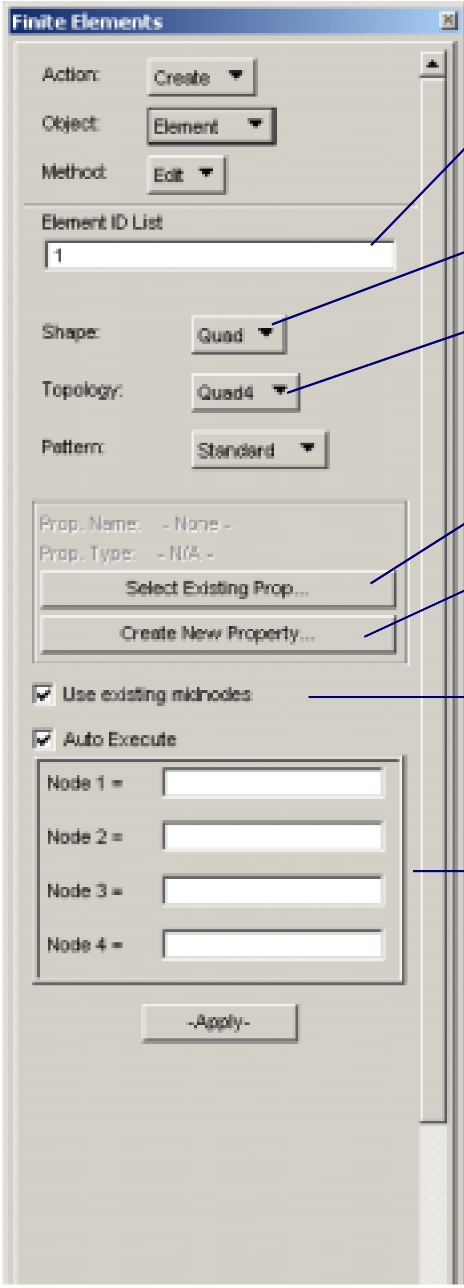
Reference Coordinate Frame

Projection Vector and Refer. Coordinate Frame is used if the Define Vector option is chosen.

Curve List/ Surface List Plane List

Depending on what you selected as the “Project onto” entity, the listbox will display the ID of the curve, surface, or plane you select for receiving the projected points.

3.3 Creating Elements



See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Select the shape of the element(s) to be created.

Choose an element topology (nodal connectivity). See **The MSC.Patran Element Library** (Ch. 15), for an illustrated list of MSC.Patran's available topology.

Use this button to select an existing property to assign to newly created elements. Once a property is selected it's name will appear in the "Prop. Name:" label, and a small description of that property type will appear in the "Prop. Type." label.

Use this button to create a property to assign to the newly created elements. Once the property is created, it is automatically selected as the property to associate to the new elements.

When selected ON, duplicate mid-edge nodes will not be created along common boundaries of elements with similar topology. Instead, existing mid-edge nodes from adjacent elements will be used when appropriate. If selected OFF, new mid-edge nodes will always be created.

Specifies lists of element corner nodes. One select databox is displayed for each element corner node. Only the elements corner nodes need to be specified. Element midnodes will be automatically generated.

Multiple elements can be created by specifying multiple corner nodes in each select databox. For example, when creating Bar elements, if the list for corner node 1 contains nodes 10 and 11, and the list for corner node 2 contains nodes 13 and 14, two elements will be created: one between nodes 10 and 13, and one between nodes11 and 14.

If the number of nodes specified in each node list are not equal, the last specified node ID in each short list will be used to extend that list to the necessary length.

3.4 Creating MPCs

Overview. An MPC (multi-point constraint) is a constraint that defines the response of one or more nodal degrees-of-freedom (called dependent degrees-of-freedom) to be a function of the response of one or more nodal degrees-of-freedom (called independent degrees-of-freedom). The general form of the MPC, which most of the major structural analysis codes support (referred to as the explicit MPC type in MSC.Patran) is as follows:

$$U_0 = C_1U_1 + C_2U_2 + C_3U_3 + ... + C_nU_n + C_0 \qquad \text{Eq. 3-1}$$

Where U_0 is the dependent degree-of-freedom, U_i the independent degrees-of-freedom, and C_i the constants. The term to the left of the equal sign is called the dependent term and the terms to the right of the equal sign are called the independent terms. C_0 is a special independent term called the constant term.

An example of an explicit MPC is:

$$UX(Node\ 4) = 0.5*UX(Node\ 5) - 0.5*UY(Node\ 10) + 1.0 \qquad \text{Eq. 3-2}$$

which specifies that the x displacement of node 4 is equal to half the x displacement of node 5 minus half they displacement of node 10 plus 1.0. There are four terms in this example, one dependent term, two independent terms, and a constant term.

MPC Types. MPCs can be used to model certain physical phenomena that cannot be easily modeled using finite elements, such as rigid links, joints (revolutes, universal, etc.), and sliders, to name a few. MPCs can also be used to allow load transfer between incompatible meshes. However, it is not always easy to determine the explicit MPC equation that correctly represents the phenomena you are trying to model.

To help with this problem, many analysis codes provide special types of MPCs (sometimes called “implicit” MPCs) which simulate a specific phenomena with minimum user input. For example, most analysis codes support an implicit MPC type which models a rigid link, in which an independent node is rigidly tied to one or more dependent nodes. All the user is required to input are the node IDs. The analysis code internally generates the “explicit” MPCs necessary to cause the nodes to act as if they are rigidly attached.

In addition to the implicit MPC types supported by the analysis code, there are implicit MPC types supported by the analysis code translator. These are converted into “explicit” form during the translation process. This allows MSC.Patran to support more MPC types than the analysis code supports itself.

MSC.Patran supports the creation of all MPC types through the use of a single form, called **Create MPC Form (for all MPC Types Except Cyclic Symmetry and Sliding Surface)** (p. 113), with two exceptions: the Cyclic Symmetry and Sliding Surface MPC types. These two MPC types have special capabilities which require special create forms. See **Create MPC Cyclic Symmetry Form** (p. 115) and **Create MPC Sliding Surface Form** (p. 116).

Before creating an MPC, first select the type of MPC you wish to create. Once the type has been identified, MSC.Patran displays the proper form(s) to create the MPC.

A list of the MPC types which are supported by the MSC analysis codes can be found in the application module User’s Guide or application Preference Guide for the respective analysis code. You will only be able to create MPCs which are valid for the current settings of the Analysis Code and Analysis Type preferences. If the Analysis Code or Analysis Type preference is changed, all existing MPCs, which are no longer valid, are flagged as such and will not be translated. Invalid MPCs are still stored in the database and are displayed, but they cannot be modified or shown. However, they can be deleted. An invalid MPC can be made valid again by setting the Analysis Code and Analysis Type preferences back to the settings under which the MPC was originally created.

MPC Terms. The principal difference between one MPC type and the next is the number and makeup of the dependent and independent terms. A term is composed of up to four pieces of information:

1. A sequence number (used to order dependent and independent terms with respect to each other).
2. A nonzero coefficient.
3. One or more nodes.
4. One or more degrees-of-freedom.

For example, a dependent term of the explicit MPC type consists of a single node and a single degree-of-freedom, while an independent term of the explicit MPC type consists of a coefficient, a single node, and a single degree-of-freedom. As another example, the dependent and independent terms of the Rigid (fixed) MPC type consist of a single node.

The number of dependent and independent terms required or allowed varies from one MPC type to the next. For example, the Explicit MPC type allows only one dependent term while allowing an unlimited number of independent terms. Conversely, the Rigid (fixed) MPC type allows one independent term while allowing an unlimited number of dependent terms. Other MPC types allow only one dependent and one independent term, or one dependent and two independent terms.

Degrees-of-Freedom. Whenever one or more degrees-of-freedom are expected for an MPC term, a listbox containing the valid degrees-of-freedom is displayed on the form. A degree-of-freedom is valid if:

- It is valid for the current Analysis Code Preference.
- It is valid for the current Analysis Type Preference.
- It is valid for the selected MPC type.

In most cases, all degrees-of-freedom which are valid for the current Analysis Code and Analysis Type preferences are valid for the MPC type. There are some cases, however, when only a subset of the valid degrees-of-freedom are allowed for an MPC. For example, an MPC may allow the user to select only translational degrees-of-freedom even though rotational degrees are valid for the Analysis Code and Analysis Type preference.

Important:	Some MPC types are valid for more than one Analysis Code or Analysis Type preference combination.

The degrees-of-freedom which are valid for each Analysis Code and Analysis Type Preference are listed in the analysis code or analysis code translator User’s Guide.

Important:	Care must be taken to make sure that a degree-of-freedom that is selected for an MPC actually exists at the nodes. For example, a node that is attached only to solid structural elements will not have any rotational degrees-of-freedom. However, MSC.Patran will allow you to select rotational degrees-of-freedom at this node when defining an MPC.

Graphics. MPCs are displayed as a set of lines which connect each dependent node (node appearing as part of a dependent term) to each independent node (node appearing as part of an independent term). The dependent nodes are circled to distinguish them from the independent nodes (see **Figure 3-1**). MPCs are treated like elements in MSC.Patran because they:

- Can be added to or removed from groups.
- Have integer IDs which can be displayed or suppressed.
- Have their own color attribute (default = red).

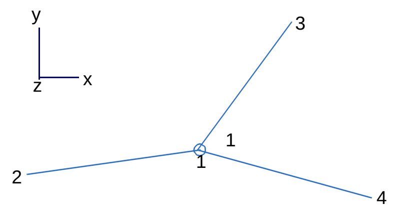


Figure 3-1 Graphical Display of an MPC with One Dependent Node and Three Independent Nodes

Creating Multiple MPCs. In certain cases, MSC.Patran allows you to create several multi-point constraints (called Sub-MPCs) at one time which are stored as a single MPC entity with a single ID. The following rules apply:

- When an MPC requires only a single node to be specified for both dependent and independent terms, you can specify more than one node per term, as long as the same number of nodes is specified in each term. The number of Sub-MPCs that will be created is equal to the number of nodes in each term. The first node in each term is extracted to define the first Sub-MPC, the second node in each term is extracted to define the second Sub-MPC, and so on.
- When an MPC requires only a single degree-of-freedom to be specified for both dependent and independent terms, you can specify more than one degree-of-freedom per term, as long as the same number of degrees-of-freedom is specified in each term. The number of Sub-MPCs that will be created is equal to the number of degrees-of-freedom in each term. The first degree-of-freedom in each term is extracted to define the first Sub-MPC, the second degree-of-freedom in each term is extracted to define the second Sub-MPC, and so on.
- When an MPC requires only a single degree-of-freedom to be specified for the dependent terms and no degrees-of-freedom for the independent terms (or vice versa), you can specify more than one degree-of-freedom per term, as long as the same number of degrees-of-freedom is specified in each term that expects a single degree-of-freedom. The number of Sub-MPCs that will be created is equal to the number of degrees-of-freedom in each term. The first degree-of-freedom in each term is extracted to define the first Sub-MPC, the second degree-of-freedom in each term is extracted to define the second Sub-MPC, and so on.

- When an MPC requires only a single node and a single degree-of-freedom to be specified for both dependent and independent terms, you can specify more than one node and/or degree-of-freedom per term, as long as the same number of nodes and degrees-of-freedom are specified in each term. The number of Sub-MPCs that will be created is equal to the number of nodes times the number of degrees-of-freedom in each term.
- For all other MPC types which do not match one of the above conditions a single Sub-MPC will be created.

When multiple Sub-MPCs are created, they are displayed as shown in **Figure 3-2**, with the ID of the MPC displayed at the centroid of each Sub-MPC. The translators will treat each Sub-MPC as a separate MPC, but MSC.Patran treats the collection of Sub-MPCs as a single entity.

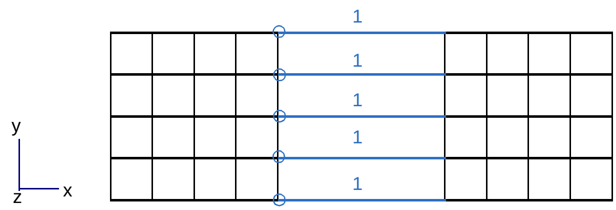
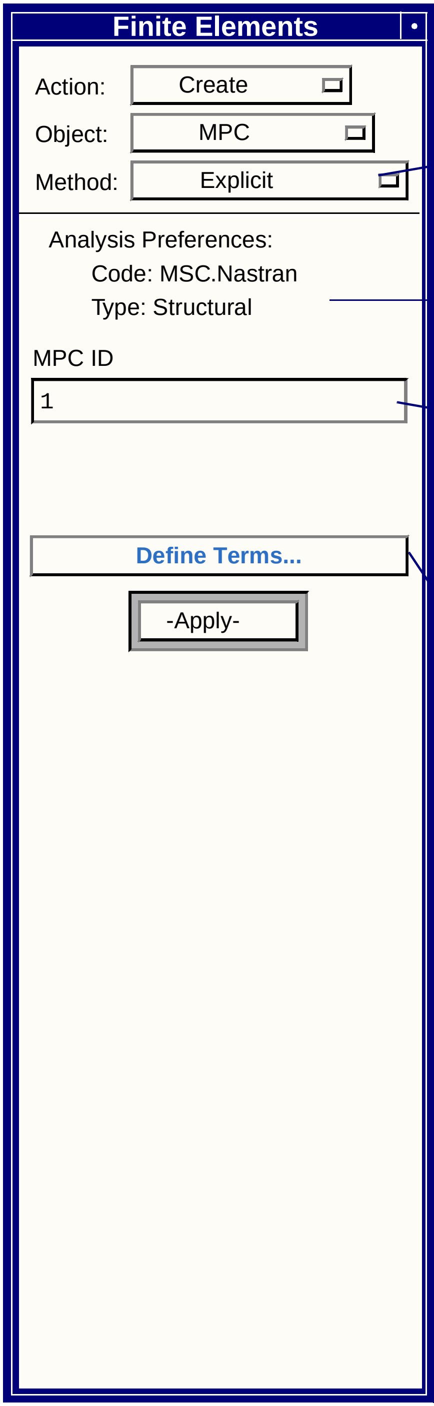


Figure 3-2 The Graphical Display of an MPC Which is Made up of Five Sub-MPCs

Create MPC Form (for all MPC Types Except Cyclic Symmetry and Sliding Surface)

When *Create* is the selected Action and *MPC* is the selected Object, the Create MPC form is displayed. Several MPC types are valid under the Type option menu.



Used to select the type of MPC to create. Only the Types that are valid for the current settings of the Analysis Code and Analysis Type preferences are presented.

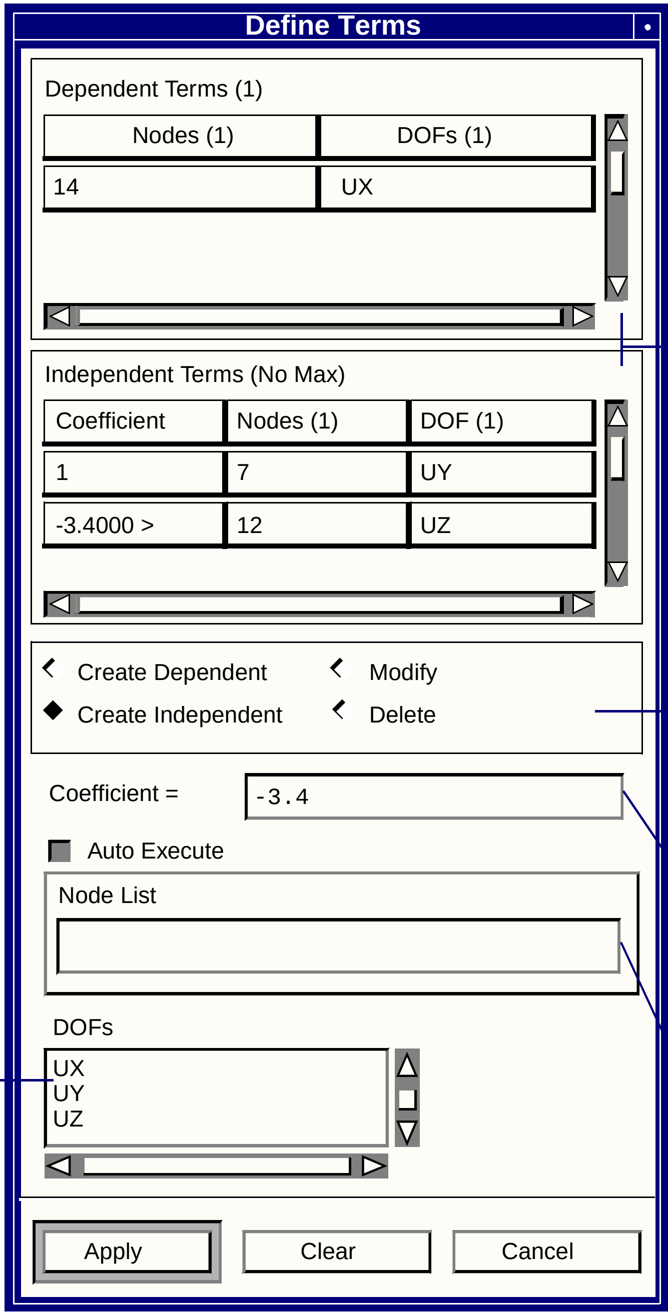
Indicates the current settings of the Analysis Code and Analysis Type Preferences.

See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*. MSC.Patran attempts to avoid MPC and element ID conflicts.

Brings up the Define Terms form. This allows you to create, modify, or delete dependent and independent terms.

Define Terms Form

The Define Terms form appears when the Define Terms button is selected on the Create MPC form. Use this form to define the dependent and independent terms of an MPC.



Holds the dependent and independent term information as rows in the spreadsheets. The number of terms required is displayed in parentheses next to the spreadsheet label. A term consists of one or more of the following:

1. A sequence number (not shown).
2. A nonzero coefficient.
3. A list of nodes (the required number is displayed in parentheses). (1*) means node ids may be entered one per term or all in one term.
4. A list of degrees-of-freedom (the required number is listed in parentheses).

Existing terms can be selected for modifications and deletion.

Sets the mode of the Apply function to **1)** create a dependent term, **2)** create an independent term, **3)** modify a term, or **4)** delete a term. The Create Dependent and Create Independent items are disabled once the maximum number of dependent or independent terms are created.

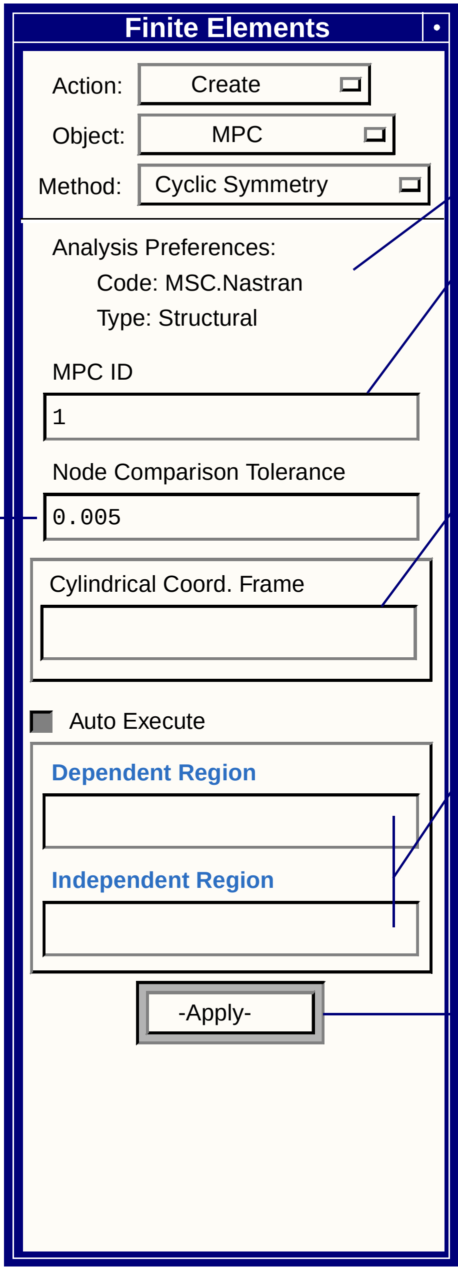
Real databox used to *specify a real nonzero coefficient for a term*. This widget is displayed when creating or modifying a term which includes a Coefficient column.

Node select databox used to *specify the nodes for a term*. This widget is displayed when creating or modifying a term which includes a Nodes column.

Listbox used to *select the degrees-of-freedom for a term*. This widget is displayed when creating or modifying a term which includes a DOFs column.

Create MPC Cyclic Symmetry Form

Use this form to Create an MPC which defines a set of cyclic symmetry boundary conditions between the nodes in two regions.



Indicates the current settings of the Analysis Code and Analysis Type Preferences.

See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Cylindrical coordinate frame whose z axis defines the axis of symmetry. This coordinate frame will be assigned as the Analysis Coordinate Frame to all nodes in the dependent and independent regions except those nodes which lie on the axis of symmetry. For these nodes, a rectangular coordinate frame whose z axis lies on the axis of symmetry will be automatically created and assigned as the Analysis Coordinate Frame.

Specifies the dependent and independent nodes on the cyclic boundaries. The Cyclic Symmetry select menu will appear to allow the user to select nodes explicitly or by reference to 2D element edges, 3D element faces, points, curves, or surfaces. The same number of unique nodes must be specified in both regions. A node can be referenced in both the dependent and independents regions only if it lies on the axis of symmetry.

When the Apply button is selected, MSC.Patran extracts the nodes associated to the entities in the dependent and independent regions and matches them up by comparing their r and z coordinates (in the specified cylindrical coordinate frame). A match must be found for every node. The following tests are then made:

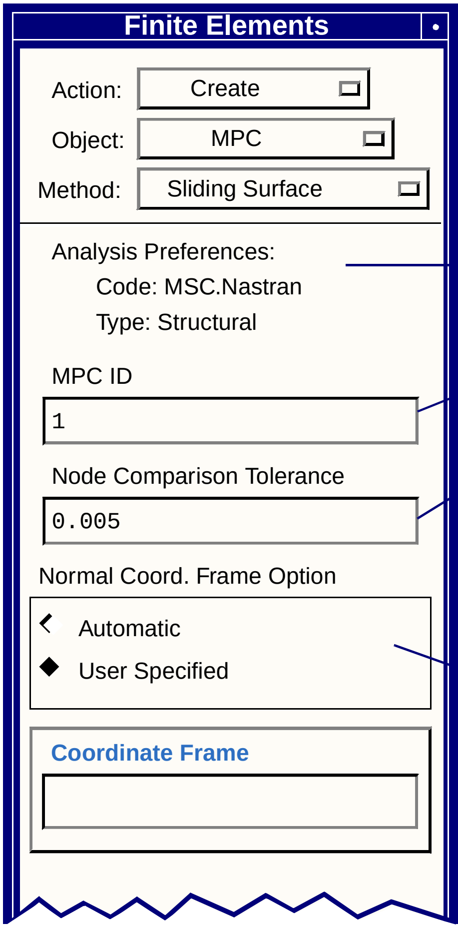
- Do all of the node pairs have the same $\Delta\theta$ (θ_i-θ_d)?
- Is the $\Delta\theta$ evenly divisible into 360 degrees?

If the answer to both of these questions is yes, an MPC will be created which ties the node pairs together. Detail on how this MPC is translated can be found in the appropriate Analysis Code Translator User's Guide.

Tolerance used to match dependent nodes with independent nodes. By default, this parameter is equal to the Global Model Tolerance set in Global Preferences. A dependent node is matched with an independent node when the r and z coordinates of the nodes (in the specified cylindrical coordinate frame) are the same within the specified tolerance.

Create MPC Sliding Surface Form

Use this form to create an MPC which defines a sliding surface between the nodes in two coincident regions. The translational degree-of-freedom (normal to the surface) of coincident nodes in the two regions are tied while all others remain free.



Indicates the current settings of the Analysis Code and Analysis Type Preferences.

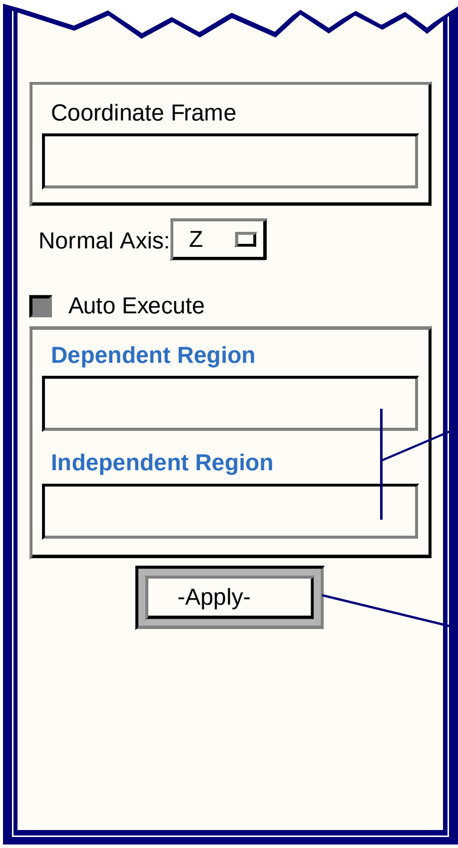
See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*1.

Tolerance used to match dependent nodes with independent nodes. By default, this parameter is equal to the Global Model Tolerance set in Global Preferences. A dependent node is matched with an independent node when the x, y, and z coordinates of the nodes are the same within the specified tolerance.

Indicates how a coordinate frame(s) with an axis normal to the surface is to be specified. The coordinate frame(s) will be assigned as the Analysis Coordinate Frame of the nodes in the dependent and independent regions:

Automatic: One or more rectangular coordinate frame(s) will be automatically created for the nodes, with the z axis defined normal to the surface. An attempt will be made to reuse coordinate frames when the normal does not change from one node to the next.

User Specified: Two additional widgets will appear to allow the user to specify both the coordinate frame and the axis, which is normal to the surface. The choices are X,Y, and Z for rectangular systems, R, T (for Theta), and Z for cylindrical systems, and R, P (for Phi), and T for spherical system.

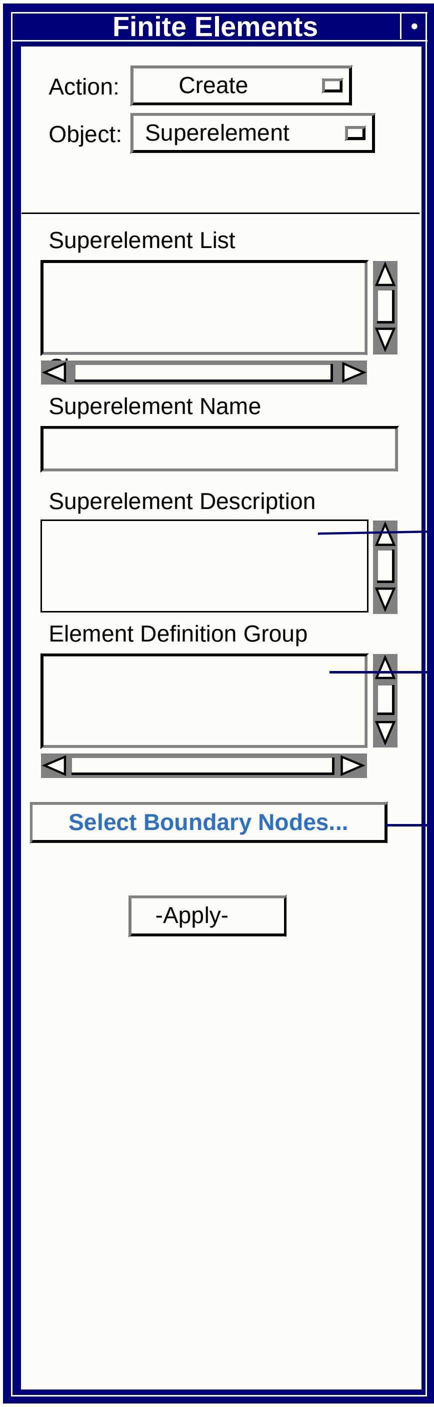


Specifies the dependent and independent nodes on the sliding surface. The Sliding Surface select menu will appear to allow the user to select nodes associated to 2D element edges, 2D elements, 3D element faces, surface edges, surfaces, and solid faces. The same number of unique nodes must be specified in both regions.

When Apply is selected, MSC.Patran extracts the nodes associated to the entities in the dependent and independent regions and matches them up by comparing their coordinates. A match must be found for every node. An MPC will be created which ties the translational degree-of-freedom which is aligned with the normal axis of each dependent node to the same degree-of-freedom of its matching independent node. Details on how this MPC is translated can be found in the appropriate Analysis Code Translator User's Guide.

3.5 Creating Superelements

This form is used to create superelements. Note that this is currently available only for the MSC.Nastran analysis preference.



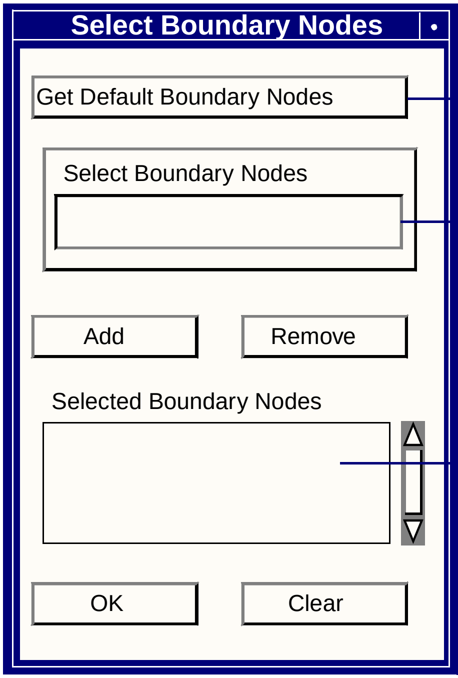
Input of superelements description. Maximum limit of 256 characters.

List of groups with elements that define superelements. If a group does not contain elements, it will not show up in the Element Definition Group listbox.

Brings up the form to define the boundary nodes. By default, the interfacing nodes between the superelement definition group and the rest of the structure are selected as the boundary nodes.

Select Boundary Nodes

When the Create Action and the Superelement Object is chosen on the finite element form and the Select Boundary Nodes is selected, the following subordinate form will appear.



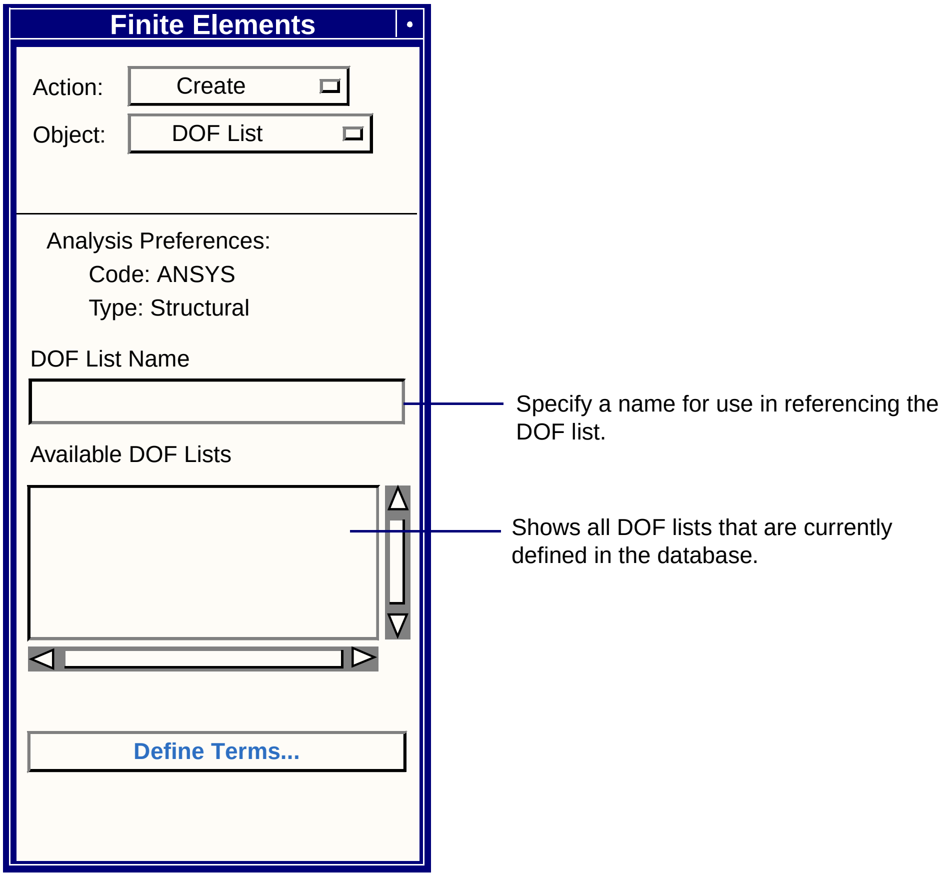
Gets the interfacing nodes between the superelement group and the rest of the structure.

May manually select the boundary nodes.

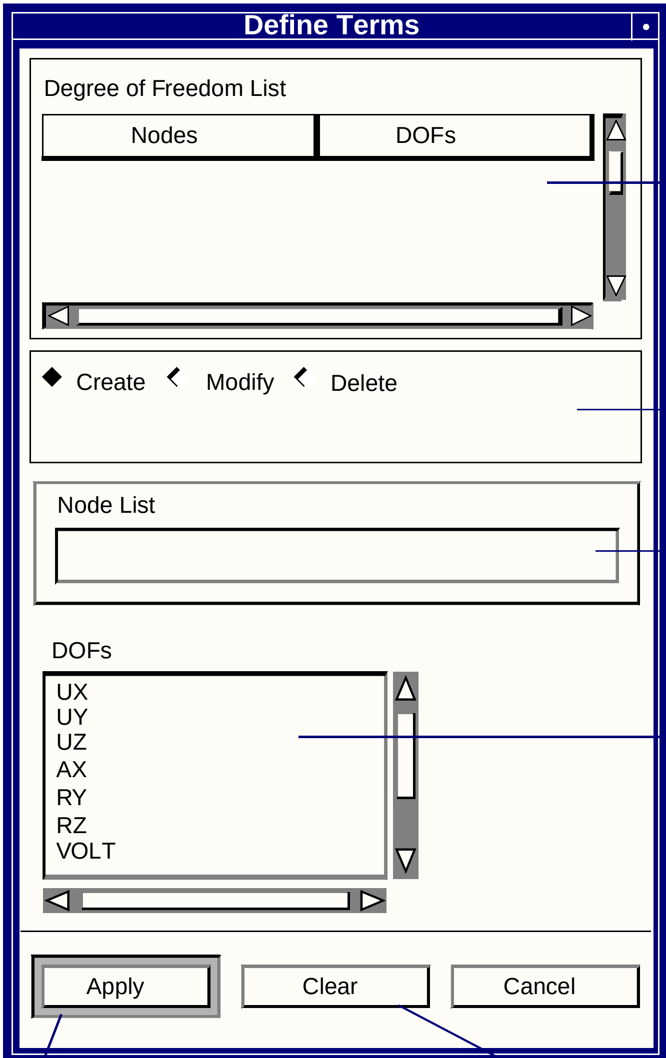
A listing of the selected boundary nodes. If left blank, the interfacing nodes between the superelement group and the rest of the structure model will be selected as boundary nodes.

3.6 Creating DOF List

Degree-of-Freedom (DOF) lists may be created by using this form and the subordinate form presented when Define Terms is selected. Note that this is currently available only for an ANSYS or ANSYS 5 analysis preference.



Define Terms



Displays the selected nodes and DOFs that were specified using the Node List databox and the DOFs listbox.

Select option to Create, Modify or Delete entries in the degree-of-freedom list.

Specify node IDs to be included in the nodes cell of the degree-of-freedom list.

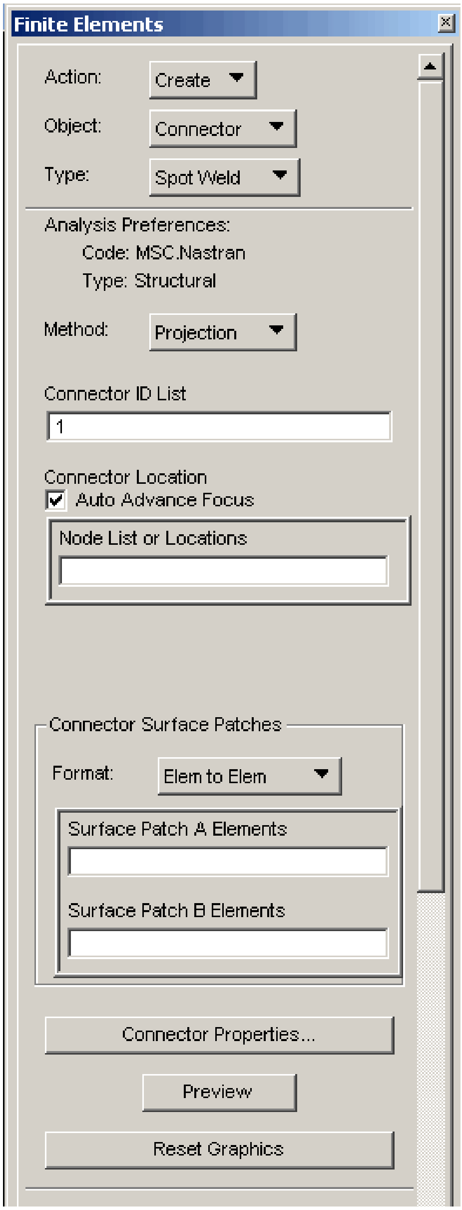
Highlight the degrees-of-freedom to be included in the DOFs cell of the degree-of-freedom list term.

For Create or Modify, selecting Apply will either place the new term into the spreadsheet or replace the selected term with the modified term. For Delete, selecting Apply will delete the selected term.

Selecting Clear will remove all terms from the degree-of-freedom list spreadsheet.

3.7 Creating Connectors

This form is used to create connectors.



Method

Select from two methods for defining the Spot Weld location:

Projection specifies a node or point in space that is to be projected onto the two surface patches of the connector to determine the end points, GA and GB.

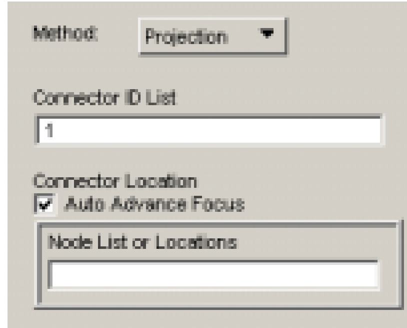
Axis specifies nodes directly for GA and GB.

Connector ID List

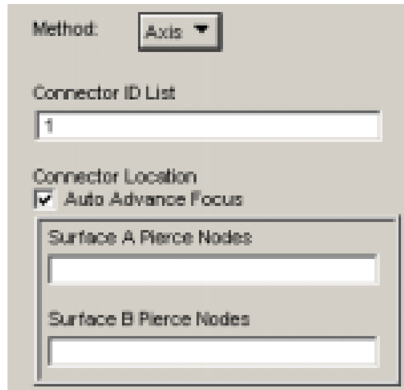
Displays the ID of the *next* connector that will be created

Connector Location

The point specified for the **Projection** method is projected onto each surface patch. Nodes are generated at those locations, and the Pierce Nodes, GA and GB, are assigned the new node IDs.



Nodes specified for the **Axis** method define the GA and GB piercing nodes directly.

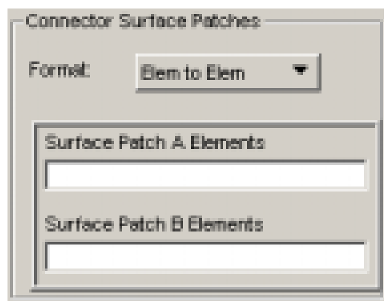


Format

Select from four formats: Elem to Elem (ELEMID and ALIGN formats, Patch to Patch (ELPAT format), Prop to Prop (PARTPAT format), and Node to Node (GRIDID format).

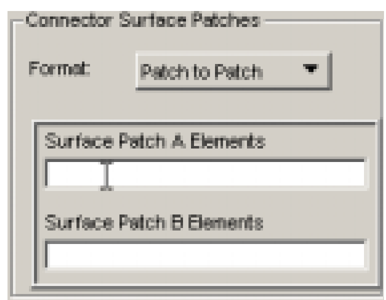
- Elem to Elem

Top and Bottom shell elements defining the surface patches for the weld. If not specified, then both GA and GB are required (ALIGN format); otherwise, one top / bottom element pair per connector is required. Regardless of GA, GB, and the weld diameter, only a single element is connected.



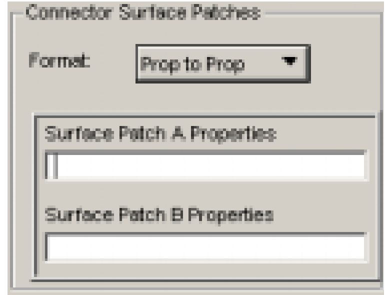
- Patch to Patch

Shell element on each surface defining the connecting surface patches (one pair per connector). Depending on the pierce locations (GA and GB) and the weld diameter, the number of connected elements may expand to up to a 3x3 element patch.



- Prop to Prop

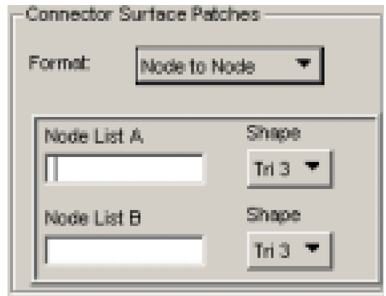
Properties associated with shell elements defining the connectivity of the weld (one pair per connector). Depending on the pierce locations (GA and GB) and the weld diameter, the number of connected elements may range from one element up to a 3x3 element patch.



Multiple connector locations may be specified for a single property pair. The same pair will be used for each connector created.

- Node to Node

Nodes (GAi and GBi) defining the connecting surface patches (one list pair per connector). The surface patches are defined as 3/6 node triangle or 4/8 node quad regions, the topology of which is indicated by SPTYP. If Node List B (GBi) is blank, then a point-to-patch connection is created.



Topology of each surface patch: Tri3, Tri6, Quad 4, Quad 8.

Connector Properties Brings up the Spot Weld Properties form (PWELD attributes).

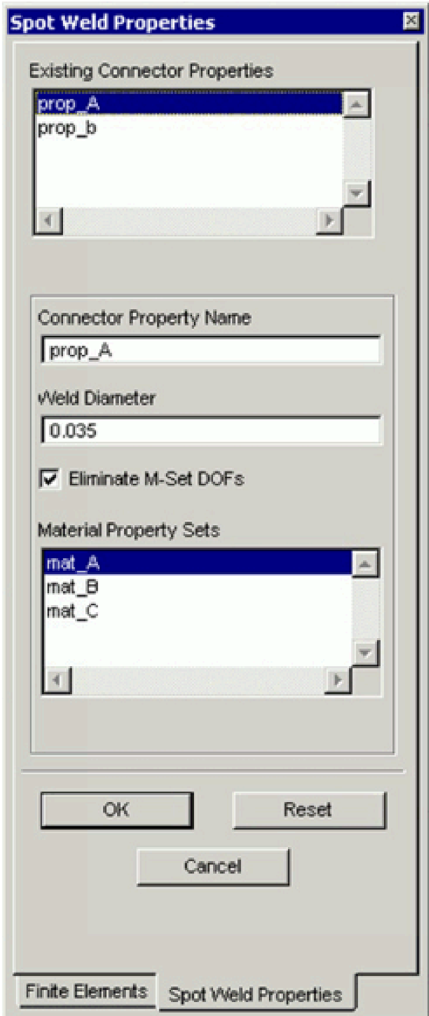
Preview Calculate/display/verify the connector (GS, GA, GB, and the connecting patches).

Spot Weld Properties Form

The Spot Weld Properties form is used to define the PWELD parameters for a Spot Weld connector.

When a new spot weld connector is created, it references a connector property, specified in this form. If that connector property does not exist in the database, one is created. If it already exists, and all the values in the existing connector property are the same as those specified here, then the existing one is referenced. If, on the other hand, it already exists, but the values are different, a warning is posted, allowing the user to overwrite the existing property, if appropriate. The default response is not to overwrite, in which case the operation is aborted.

Connector properties may be modified via the Elements / Modify / Connector form. They cannot be deleted explicitly, but will be automatically deleted when the parent connector is deleted and no other connectors reference the connector property.



Connector Property Name	The connector property name (required). Select an exsting name from the above list, or type in a new one.
Weld Diameter	The spot weld diameter (require, no default).
Eliminate M-Set DOFs	M-Set DOF elimination flag (default OFF).
Material Property Sets	The material property defining the weld material (required, no default).

Creating Fastener Connectors

The GUI for creating fasteners (CFASTs) shall be consistent with that described above for the Spot Weld Connector.

There are two primary differences:

- CFAST only has two formats, PROP and ELEM. These are analogous to PARTPAT and ELPAT of CWELD, respectively.
- Other than the diameter specification, the PFAST properties are completely different than PWELD. They are:

D	The diameter (> 0.0, required)
MCID	The element stiffness coordinate system (>= -1, default -1)
MFLAG	= 0, MCID is relative (default) = 1, MCID is absolute
KTi	Stiffness values in directions 1-3 (real, required)
KRi	Rotational stiffness values in directions 1-3 (default 0.0)
MASS	Lumped mass of the fastener (default 0.0)

CHAPTER

4

The Transform Action

- Overview of Finite Element Modeling Transform Actions
- Transforming Nodes
- Transforming Elements

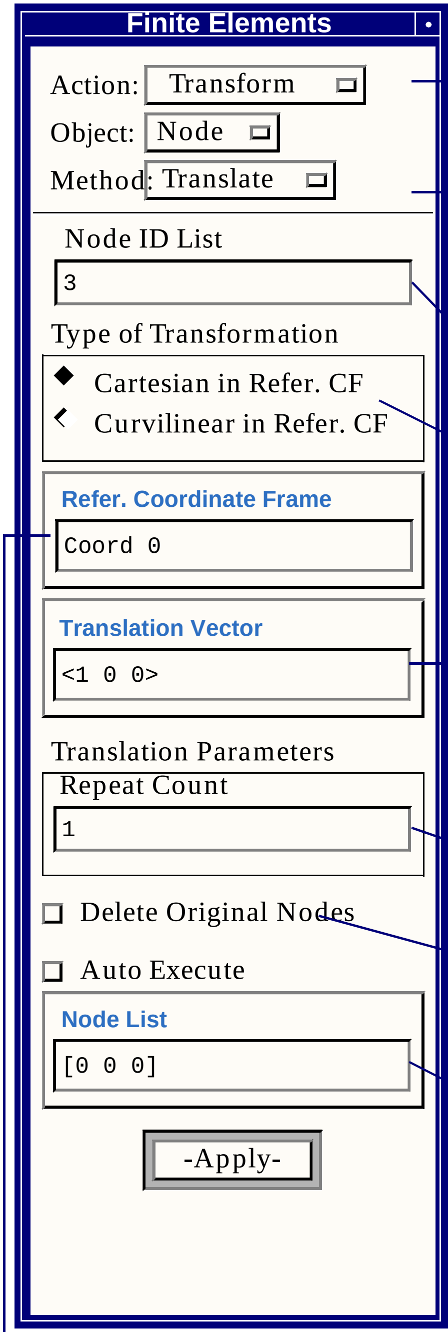
4.1 Overview of Finite Element Modeling Transform Actions

The transformations described in the following sections are identical to their counterparts in ASM. The table below lists the objects (nodes and elements) and methods that are available when *Transform* is the selected Action.

Object	Method
Node	Translate Rotate Mirror
Element	Translate Rotate Mirror

4.2 Transforming Nodes

Create Nodes by Translating Nodes



New nodes may be constructed by translating existing nodes, one or more times, as indicated by a vector. The translation performed in each iteration is:

$$\mathbf{N}_n = \mathbf{N}_{n-1} + \mathbf{T}$$

where $\mathbf{N}_n$ = the translated node location after the **nth** iteration, $\mathbf{N}_{n-1}$ = the node location prior to the **nth** iteration (in the first iteration $\mathbf{N}_{n-1}$ is the original node), and **T** = the translation vector ($\mathbf{T}_{xyz}$, $\mathbf{T}_{rqz}$, or $\mathbf{T}_{r\theta\phi}$, depending on the selected type of transformation).

See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies whether the transformation is to be performed relative to rectangular coordinates in any selected coordinate frame, or relative to curvilinear coordinates of a selected cylindrical or spherical reference coordinate frame.

Specifies the translation vector. If *curvilinear* transformation was selected, enter the vector in the coordinates of the reference coordinate system (rθz for Cylindrical, or rθϕ for Spherical). If Cartesian transformation is selected, the Vector select menu appears.

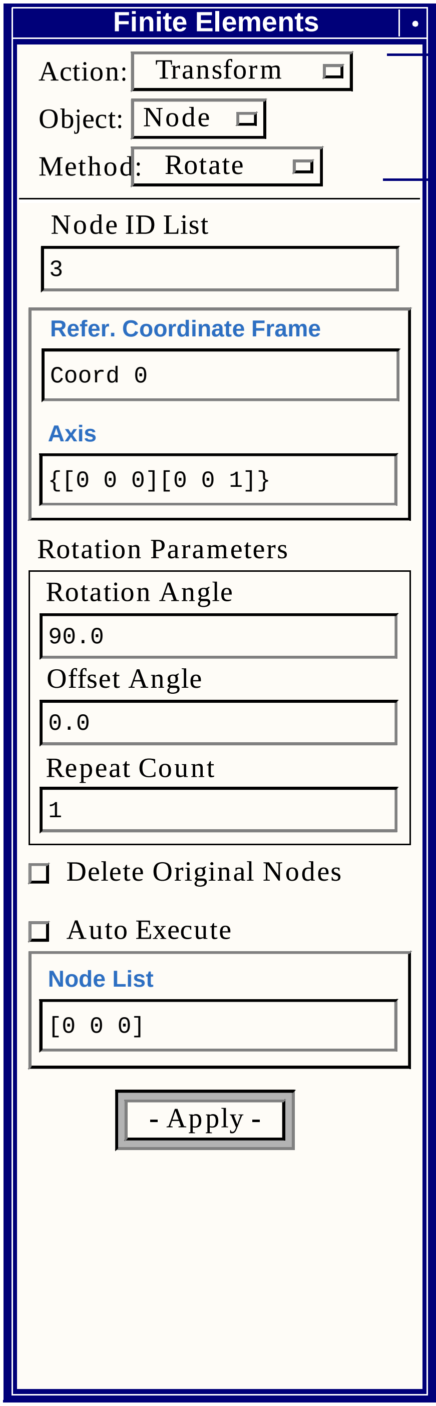
Specifies the number of times to repeat the transformation.

Turn this toggle ON to delete the original nodes or elements. This allows the original IDs to be used for the new nodes or elements.

Specifies the list of nodes to be translated.

Specifies a reference coordinate frame in which to translate the nodes. If a curvilinear transformation is desired, a cylindrical or spherical coordinate frame should be selected. The default ID is that of the default coordinate frame.

Create Nodes by Rotating Nodes

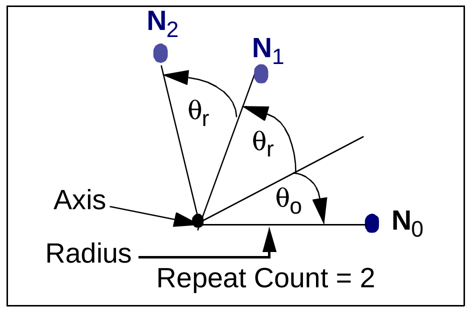


New nodes may be constructed by rotating nodes clockwise (CW) or counterclockwise (CCW) around an axis. The first rotation is specified as the sum of the Rotations Angle plus an offset. Any additional rotations (as specified by the Repeat Count parameter) are by the rotation angle only. The transformation performed for each iteration is:

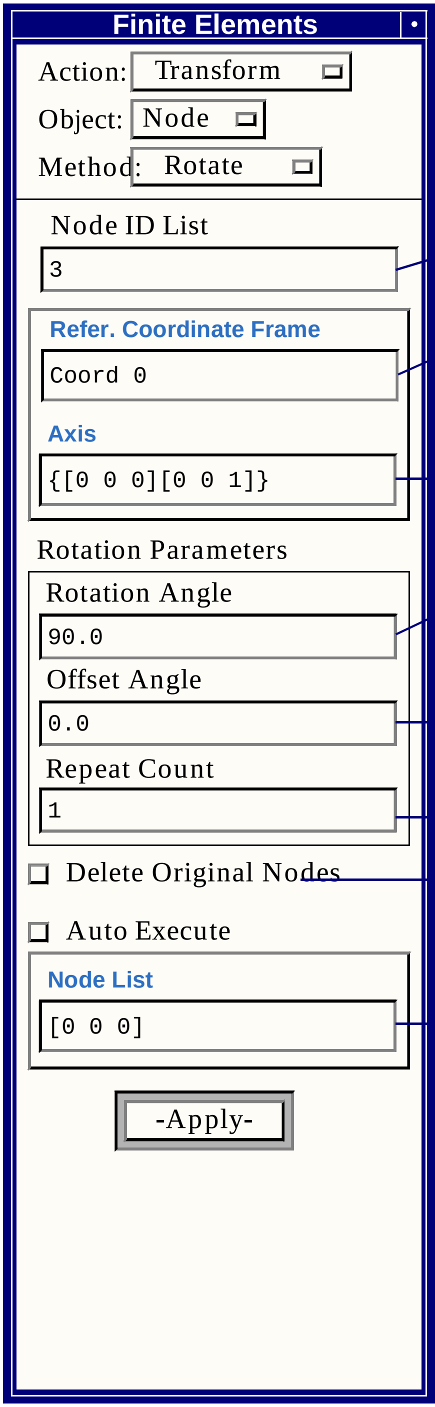
$$\text{1st Iteration: } \mathbf{N_1} = \mathbf{N_0} + \mathbf{R}(\theta_o + \theta_r)$$

$$\text{Iterations 2 \& up: } \mathbf{N_n} = \mathbf{N_{n-1}} + \mathbf{R}(\theta_{n-1} + \theta_r)$$

where $\mathbf{N_0}$ = the original node, $\mathbf{N_1}$ = the rotated location of node $\mathbf{N_0}$ after the first iteration, $\mathbf{R}$ = the radius of rotation, θ_o = the offset angle, θ_r = the rotation angle, $\mathbf{N_n}$ = the rotated location of a node after the **nth** iteration, $\mathbf{N_{n-1}}$ = the node location prior to the **nth** iteration, and θ_{n-1} = the angular displacement of node $\mathbf{N_{n-1}}$ from the original node, $\mathbf{N_0}$.



The rotation plane is established by the axis and the node to rotate. The axis is a vector that is normal to the plane of rotation. The radius of rotation, a straight line in the plane of rotation, extends from the original node, $\mathbf{N_0}$, to the point of intersection with the projected extension of the axis.



See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies a reference coordinate frame in which to rotate the nodes. The default ID is that of the default coordinate frame. The Frame select menu appears.

Defines a vector normal to the plane of rotation. The Axis select menu appears.

Specifies the rotation angle, θ_r (in degrees) through which the nodes are to be rotated, default is 90 degrees CCW. Enter a negative angle if the rotation is to be CW.

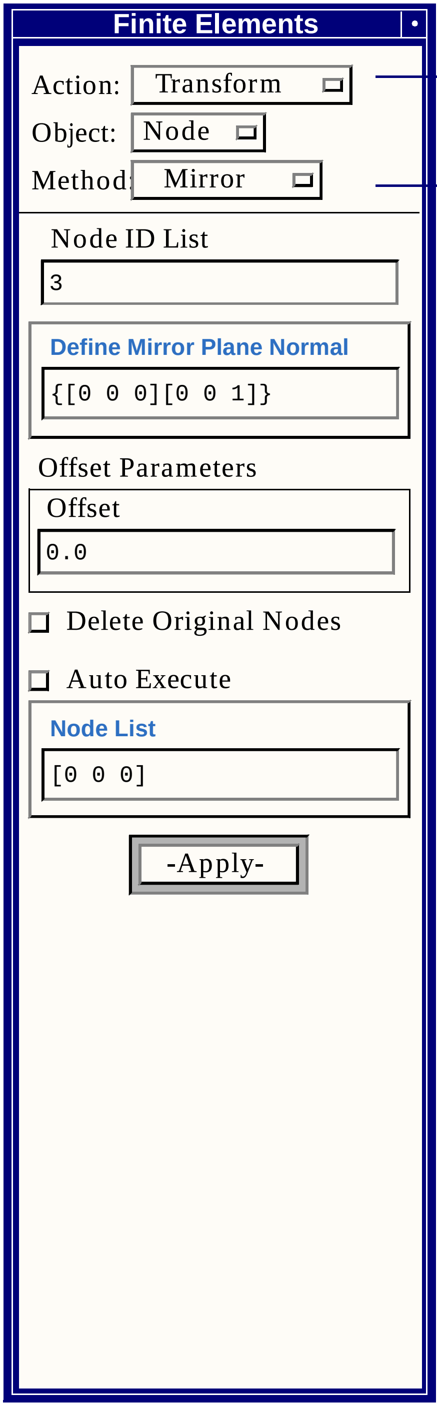
Specifies the angular offset, θ_o, (in degrees), default is 0. Enter a negative angle if the offset is to be CCW.

Specifies the number of times to repeat the transformation.

Turn this toggle ON to delete the original nodes or elements. This allows the original IDs to be used for the new nodes or elements.

Specifies the list of nodes to be rotated. The Node Select filter is in effect.

Create Nodes by Mirroring Nodes

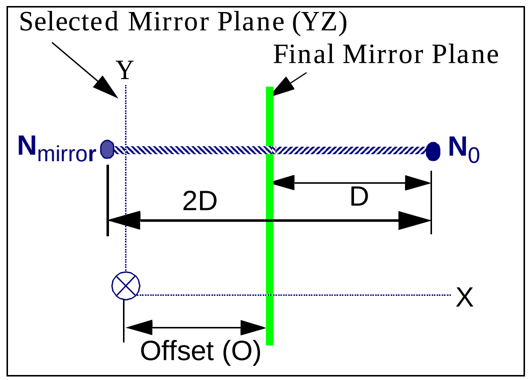


Constructs new nodes by reflecting the listed nodes about a final mirror plane, that may be offset from a selected mirror plane. The transformation performed is as follows:

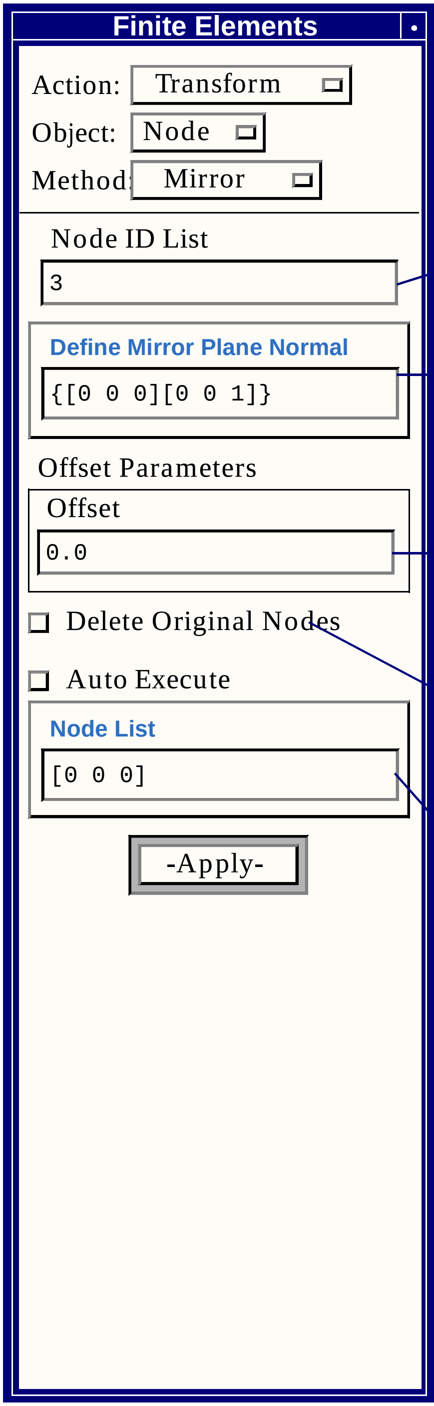
$$\mathbf{N}_{mirror} = \mathbf{N}_0 - (2D)\mathbf{n}$$

where $\mathbf{N}_{mirror}$ = the location of the new node, and D = the distance (in the direction of vector **n**, which is normal to the final mirror plane) from the final mirror plane to the original node, $\mathbf{N}_0$.

The selected mirror plane can be any arbitrary plane in model space. If a nonzero offset, O, is specified, the final mirror plane used in the mirror action is offset from the selected mirror plane by the distance O in the direction of vector **n**.



In the example illustrated above, the selected mirror plane is the YZ plane of the global Cartesian coordinate system. Therefore, the offset and distance D are measured along the X axis (vector **n**), which is normal to the YZ plane.



See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies the normal to the plane that will serve as the selected mirror plane. The Axis select menu appears. The normal to the plane is defined by a vector, where the vector base originates in the plane, and the vector tip is normal to the plane.

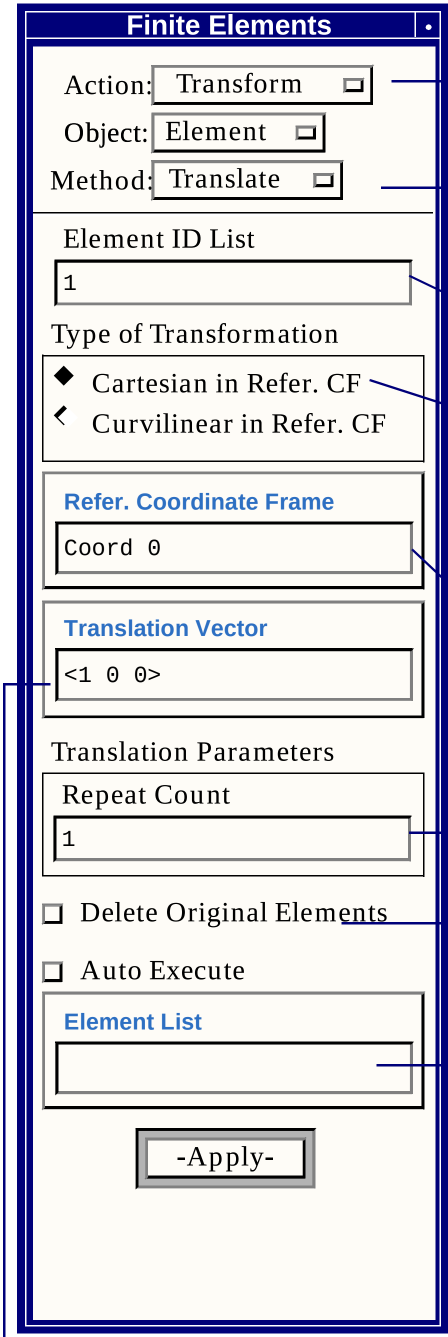
Specifies offset of final mirror plane from selected mirror plane specified above. The offset is the distance (positive or negative), along the normal, from the selected mirror plane to the location of the final mirror plane.

Turn this toggle ON to delete the original nodes or elements. This allows the original IDs to be used for the new nodes or elements.

Specifies the list of nodes to be mirrored. The Node Select filter is in effect.

4.3 Transforming Elements

Create Elements by Translating Elements



Constructs new elements by performing a rigid-body or curvilinear (nonrigid body) translation, of each elements' nodes, one or more times by an amount specified by a translation vector (number of iterations is determined by the Repeat Count).

An element is transformed by transforming each of its nodes. The result is exactly the same as if each of the nodes had been individually transformed.

See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies whether the transformation is to be performed relative to rectangular coordinates in any selected coordinate frame, or relative to curvilinear coordinates of a selected cylindrical or spherical reference coordinate frame.

Specifies a reference coordinate frame in which to translate the elements. If a curvilinear transformation is desired, a cylindrical or spherical coordinate frame should be selected. The default ID is the same as the default coordinate frame.

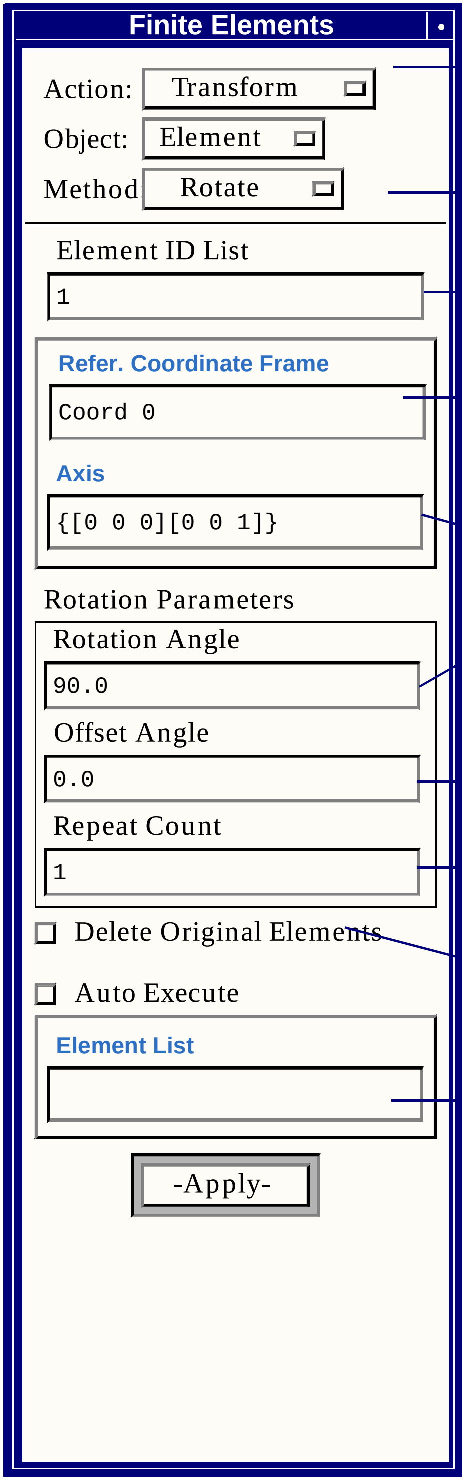
Specifies the number of times to repeat the transformation.

Turn this toggle ON to delete the original nodes or elements. This allows the original IDs to be used for the new nodes or elements.

Specifies the list of elements to be translated. The Element select menu appears.

Specifies the translation vector. If *curvilinear* transformation was selected, enter the vector in the coordinates of the reference coordinate system (rθz for Cylindrical, or rθφ for Spherical). If Cartesian transformation is selected, the Vector select menu appears.

Create Elements by Rotating Elements



Constructs new elements by performing a rigid-body rotation of each elements' nodes one or more times by an amount specified by a rotation angle (number of iterations is determined by the Repeat Count).

An element is transformed by transforming each of its nodes. The result is exactly the same as if each of the nodes had been individually transformed.

See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies a reference coordinate frame in which to rotate the elements. The default ID is that of the default coordinate frame. The Frame select menu appears.

Defines a vector normal to the plane of rotation. The Axis select menu appears.

Specifies the rotation angle, θ_r (in degrees), through which the elements are to be rotated default is 90 degrees CCW. Enter a negative angle if the rotation is to be CW.

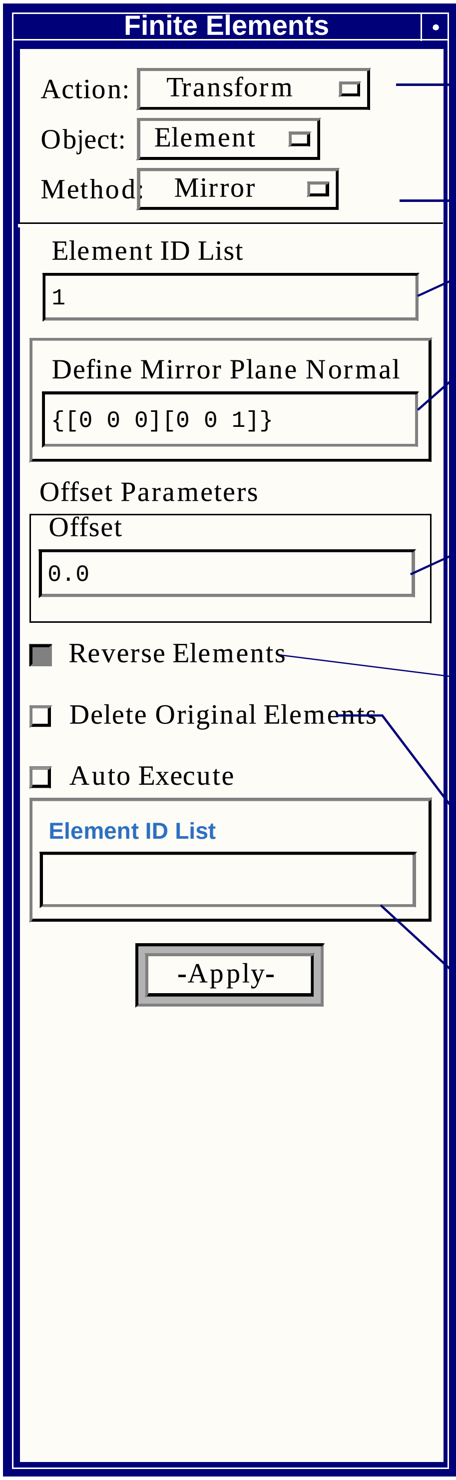
Specifies the angular offset, θ_o (in degrees), default is 0. Enter a negative angle if the offset is to be CCW.

Specifies the number of times to repeat the transformation.

Turn this toggle ON to delete the original nodes or elements. This allows the original IDs to be used for the new nodes or elements.

Specifies the list of elements to be rotated. The Element select menu appears.

Create Elements by Mirroring Elements



Constructs new elements by reflecting each element's nodes about a final mirror plane that may be offset form a selected mirror plane.

An element is transformed by transforming each of its nodes. The result is exactly the same as if each of the nodes had been individually transformed.

See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies the normal to the plane that is to serve as the selected mirror plane. Axis select menu appears. The normal to the plane is defined by a vector where the vector base originates in the plane and the vector tip is normal to the plane.

Specifies offset of final mirror plane from selected mirror plane specified above. The offset is the distance (positive or negative), along the normal, from the selected mirror plane to the location of the final mirror plane.

If ON, the element connectivity will be reversed. Reversing elements ensures that shell elements have consistent normals, and that solid elements have positive volumes. (Note: Negative volume elements will not be created under any circumstances.)

Turn this toggle ON to delete the original nodes or elements. This allows the original IDs to be used for the new nodes or elements.

Specifies the list of elements to be mirrored. The Element select menu appears.

CHAPTER 5

The Sweep Action

- Introduction
- Sweep Forms

5.1 Introduction

Sweeping elements is the process of creating higher order elements by sweeping a lower order element through a prescribed path. Therefore, a hex element may be created by sweeping a quad element through space, the edges of the hex being defined by the corners of the quad as its nodes move along the path. Ten methods for defining the swept paths are provided: Arc, Extrude, Glide, Glide-Guide, Normal, Radial Cylindrical, Radial Spherical, Spherical Theta, Vector Field and Loft.

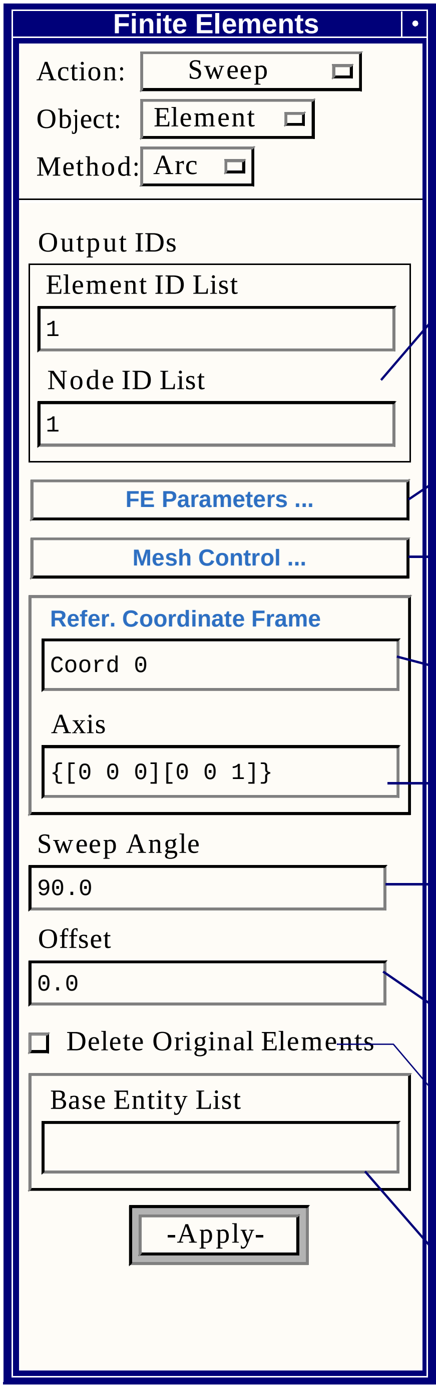
5.2 Sweep Forms

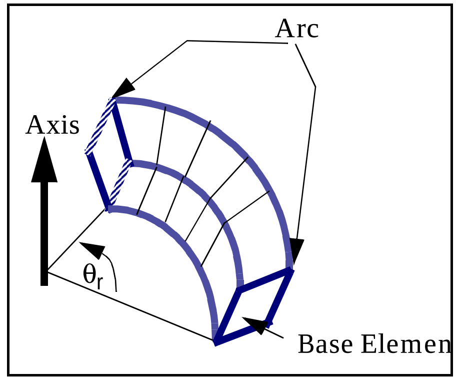
The following options are available when *Sweep* is the selected Action and *Element* is the selected Object.

Method	Description
Arc	The Arc method allows the creation of one or more elements by sweeping a surface element about an axis of rotation.
Extrude	The Extrude method allows creation of one or more elements by moving a base element through space along a defined vector.
Glide	The Glide method allows the creation of one or more elements by sweeping the base element along the path of a glide curve.
Glide-Guide	The Glide-Guide method allows the creation of one or more elements by sweeping the base element along the path of a glide curve, while the orientation with respect to the base is determined by means of a guide curve.
Normal	The Normal method allows creation of one or more elements by sweeping a base of element in a normal direction.
Radial Cylindrical	The Radial Cylindrical method allows creation of one or more elements by sweeping the base element through space radially outward from a center axis.
Radial Spherical	The Radial Spherical method allows creation of one or more elements by sweeping the base element through space radially outward from a center point.
Spherical Theta	The Spherical Theta method allows creation of one or more elements by sweeping the base element through space along a path on a sphere that is like sweeping in the latitude direction in the earth's latitude and longitude system.
Vector Field	The Vector Field method allows creation of one or more elements by sweeping a base element in a direction as determined by evaluating a vector field at each of the base nodes.
Loft	The Loft method allows creation of one or more elements by sweeping a 2D base element to the location of a 2D top element. The two meshes have to be topological congruent.

The Arc Method

The *Arc method* allows the creation of one or more elements by sweeping base entities about an axis of rotation, as shown below. The element edge length in the swept direction is defined explicitly, similar to creating a mesh seed for the meshing function.





Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for elements and nodes

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the *Reference Coordinate Frame* where the Axis Vector is defined. The default ID is that of the active coordinate frame.

Specifies the axis of rotation by defining a vector normal to the plane of rotation.

Specifies a positive or negative rotation angle (θ_r), in degrees, through which the elements are to be swept. Direction of the rotation is determined by the right hand rule. Negative angles may be specified.

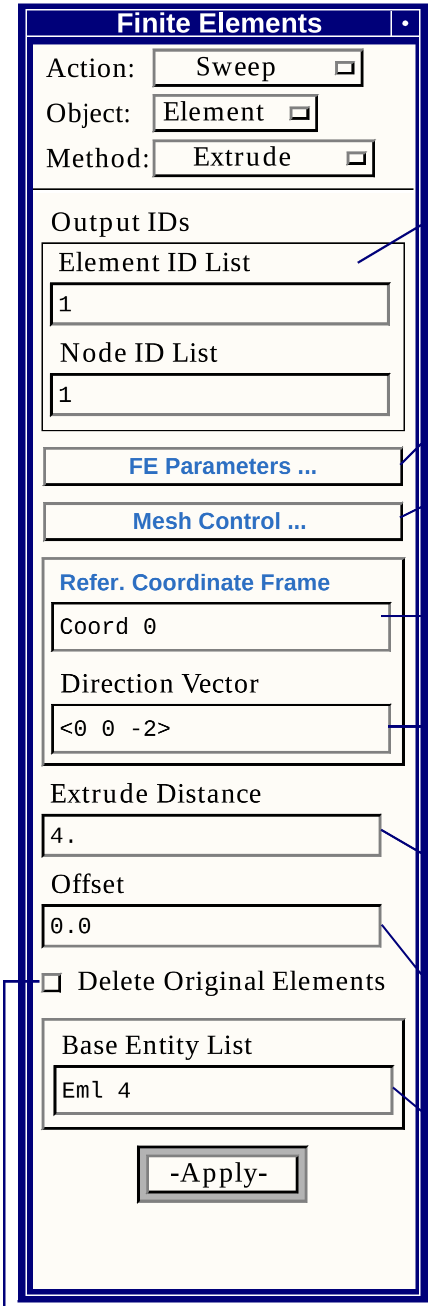
Specifies an angular offset, in degrees, for the initial base locations. Negative offset angles may be specified.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of base entities to be swept.

The Extrude Method

The Extrude method allows creation of one or more elements by moving a base entities through space along a defined vector. The extrusion vector is applied to each listed entity.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the *Reference Coordinate Frame* in which the Direction Vector is defined. The default ID is the same as the active coordinate frame.

Specifies the direction of the translation. When the direction vector is changed, the Magnitude databox is automatically loaded with the corresponding magnitude of the new vector.

Specifies the distance to extrude along the direction vector. This value is automatically loaded when the direction vector is changed. If negative the sweep will occur in the opposite direction of the direction vector.

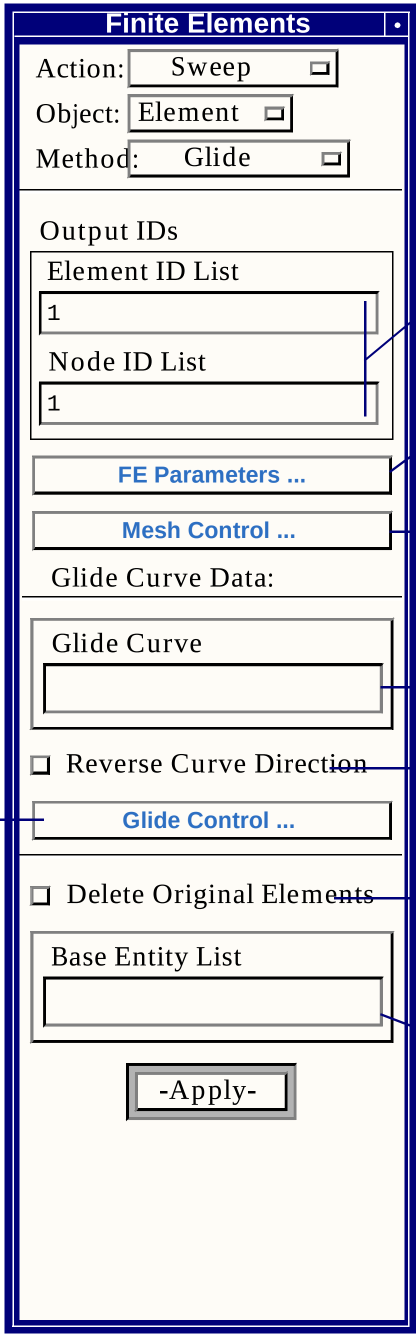
Specifies an offset for the initial base locations. Negative offsets may be specified.

Specifies a list of entities which are to be swept.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

The Glide Method

The Glide method allows the creation of one or more elements by sweeping the base element along a portion or all of a glide curve. The glide curve can exist anywhere in the model and can be traversed in either direction.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the curve along which to sweep the base entities.

Toggle ON to cause the sweep to start at the end of the curve and go toward the start.

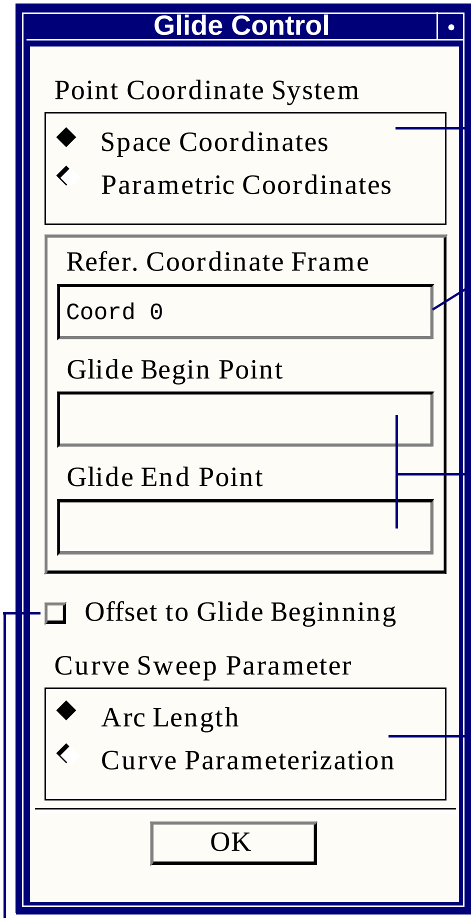
Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

Brings up the Glide Control form. Use to specify beginning and ending sweep locations along the curve and to set the curve sweep parameter as being in arc space or parametric space.

Glide Control

The Glide Control allows curves in the model to be used without having to perform simple operations such as break and translate. It also allows for sweeping to be done in arc length or parametric coordinates along the curve.



Toggles whether space or parametric coordinate system is used to specify the beginning and ending points below.

Specifies the *Reference Coordinate Frame* in which the *Beginning* and *Ending Points* are defined. The default ID is the same as the active coordinate frame. This is only used when *Space Coordinates* is the selected system.

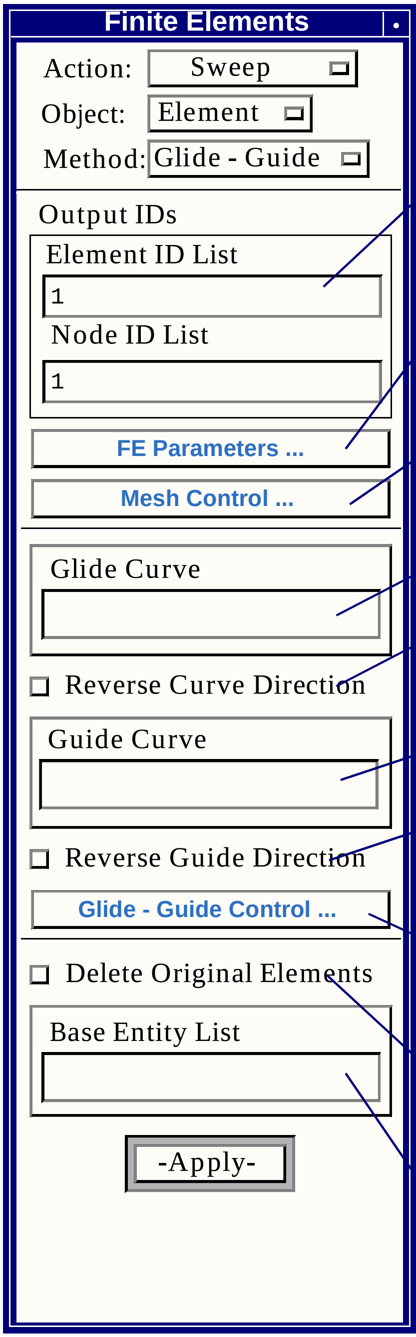
Beginning and Ending points on curve along which to do the sweep. If the Parametric Coordinates are toggled on above, these are entered as parametric values on the curve. Points can be in any order along curve. Beginning and ending points on curve are used by default if left blank.

Toggle Curve Sweep Parameter. Specifies whether mesh seeding is done in units of arc length or curve parameterization.

Offset from begin/end of curve to begin point. Swept mesh begins at begin point and proceeds to the end point.

The Glide-Guide Method

The Glide-Guide method allows the creation of one or more elements by sweeping the base element along the path of a glide curve, while the orientation with respect to the base is determined by means of a guide curve. The sweep offset is determined by the glide curve. The orientation is determined by the glide curve tangent direction and the direction to the guide curve.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the curve along which to sweep the base entities.

Toggle ON to cause the sweep to occur in the reverse of the glide curve direction. Should be used when the end of the curve is at the beginning of the desired sweep.

Specifies the guide curve to use for determining the orientation of the swept entities.

Toggle ON to cause the sweep to occur in the reverse of the guide curve direction. Should be used when the end of the curve is at the beginning of the desired sweep. Note that reversing the direction does not mirror it.

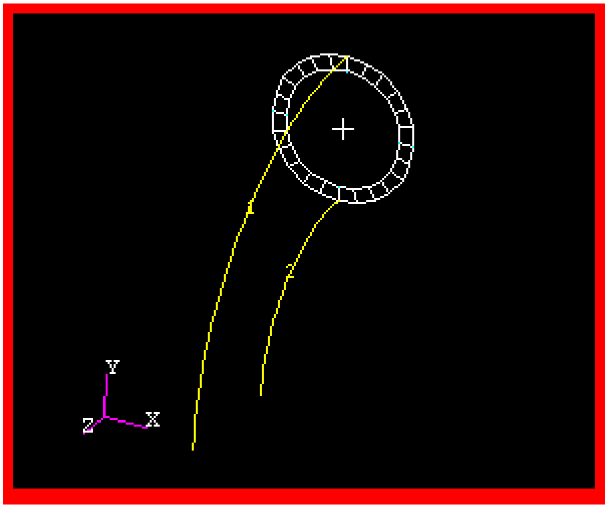
Brings up the Glide-Guide Control form. Use to specify beginning and ending sweep locations along the glide and guide curves and to set the curve sweep parameter as being in arc space or parametric space.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

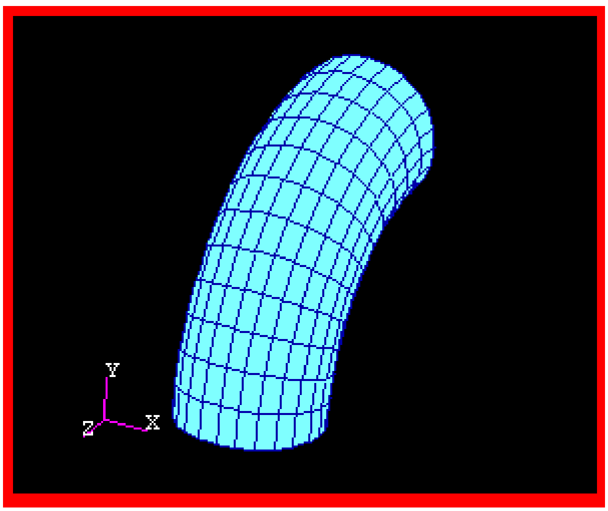
The Glide-Guide method allows sweeps to be swept and rotated along a desired path. One good application of this method is that of meshing a pipe as it goes around a bend.

Before:



For this example, the base entities are a quad mesh on the cross section of a pipe. The glide curve is curve 1 and the guide curve is curve 2. The glide curve can be thought of as the top seam of the pipe. In this case, the guide curve is along the bottom seam of the pipe. The centerline of the pipe could just as easily been used.

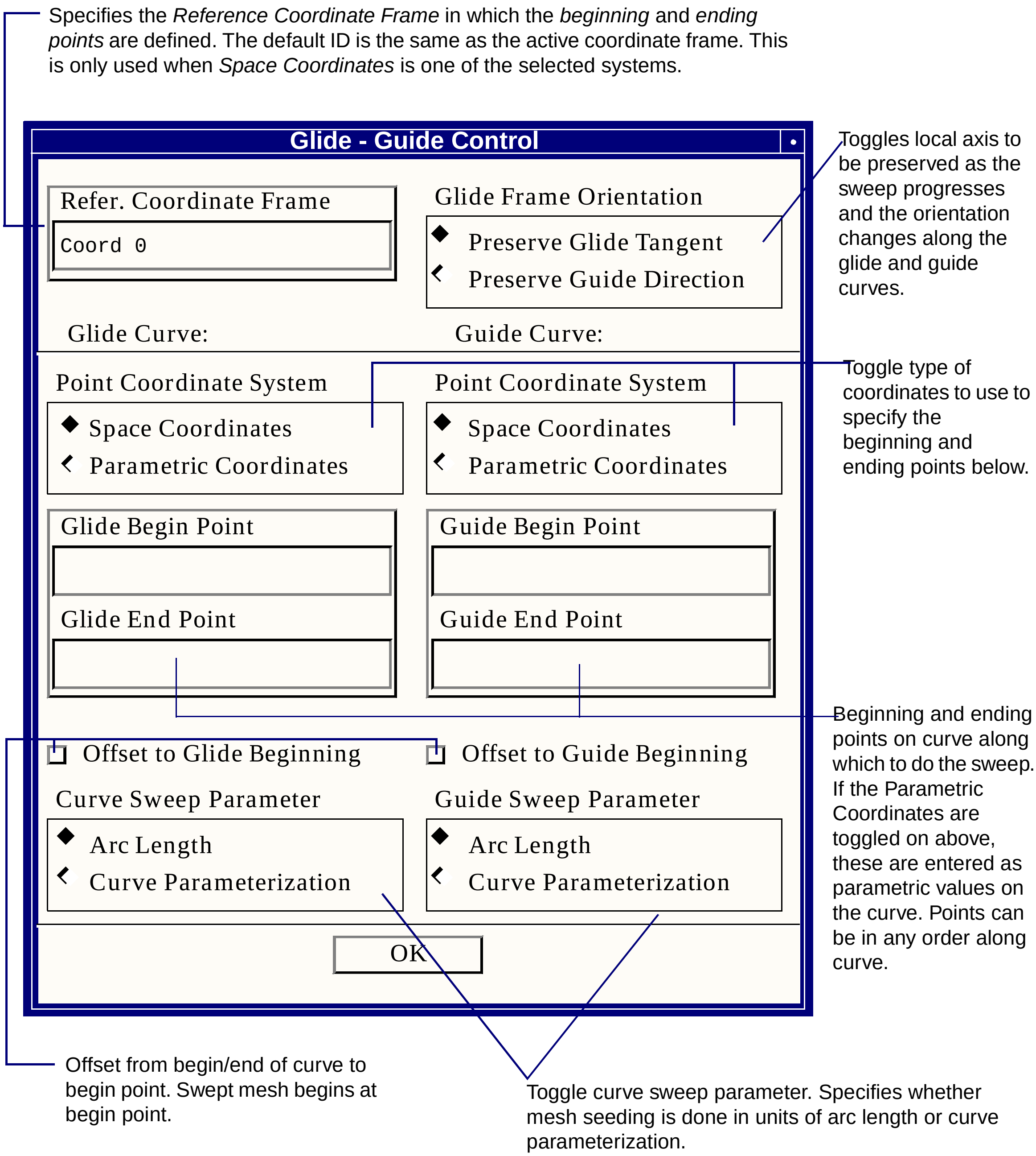
After:



Although only a 90-degree bend is shown here, the glide and guide curves are not limited to a specified angle or number of bends.

Glide-Guide Control

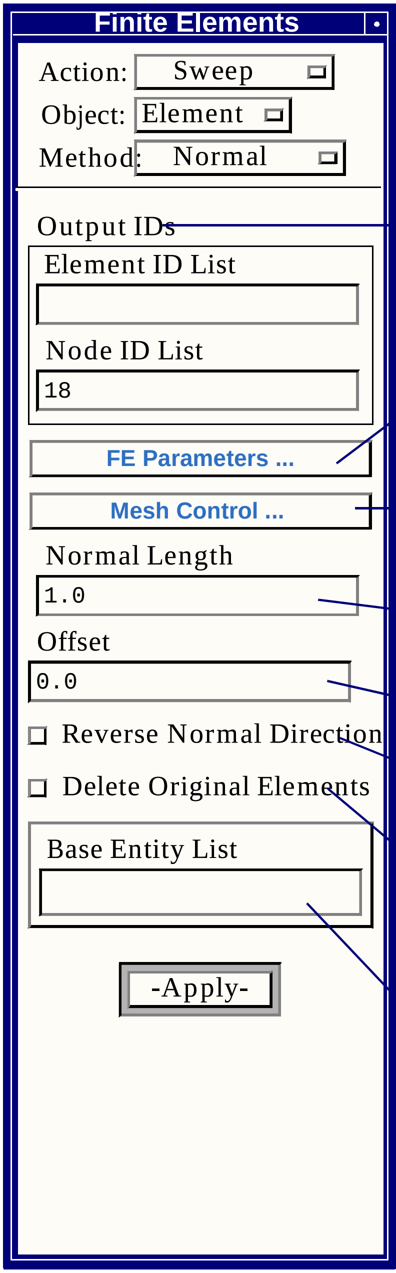
The Glide-Guide Control allows curves in the model to be used without having to perform simple operations such as break and translate. It also allows for sweeping to be done in arc length or parametric coordinates along the curve. Note that for Glide-Guide, the beginning or end of the curves should touch the base elements for best results. Otherwise, undesirable results may occur due to the large effect of orientation's rotations on the base entities.



The Normal Method

The Normal method allows creation of one or more elements by sweeping base entities in a normal direction. If the base elements are associated with geometry, the normal direction for each node will be the surface normal at that location. If the elements are unassociated, the normal direction will be the average of the element normals of all the elements in the base entity list referencing the node.

For unassociated base elements, the normals must be consistent. If not, an error is reported and execution terminated.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for the nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the sweep distance. If negative the sweep will occur in the opposite direction of the normal vector.

Specifies an offset for the initial base locations. Negative offsets may be specified.

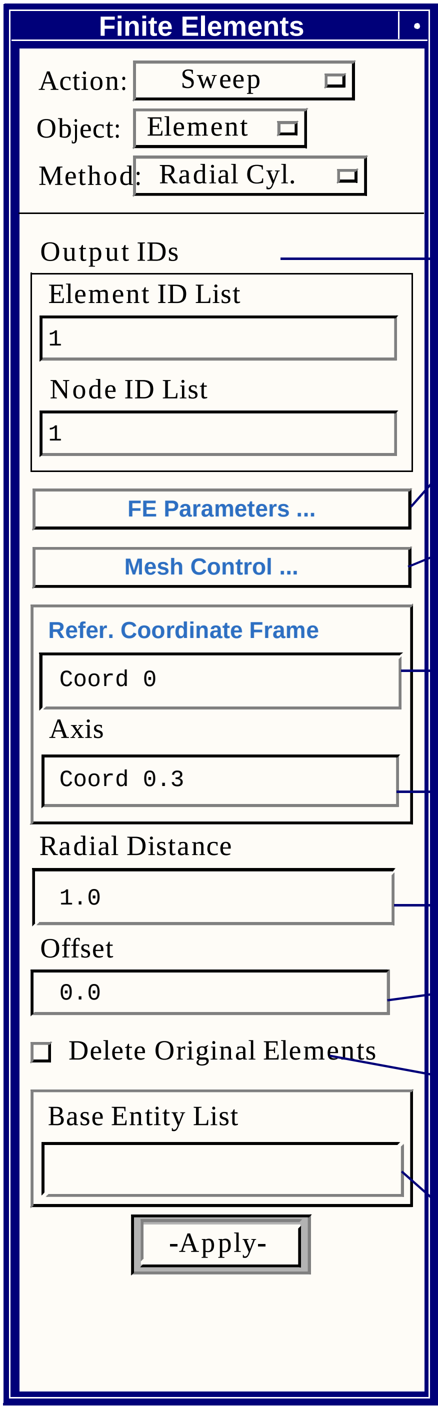
Toggle ON to cause the sweep to occur in the opposite direction indicated by the sign of the *Normal Length* parameter.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

The Radial Cylindrical Method

The Radial Cylindrical method allows creation of one or more elements by sweeping the base element through space radially outward from a center axis.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the *Reference Coordinate Frame* in which the Axis is defined. The default ID is the same as the active coordinate frame.

Specifies the cylindrical axis. All elements are swept out from this central axis.

Total distance to sweep in radial direction from axis.

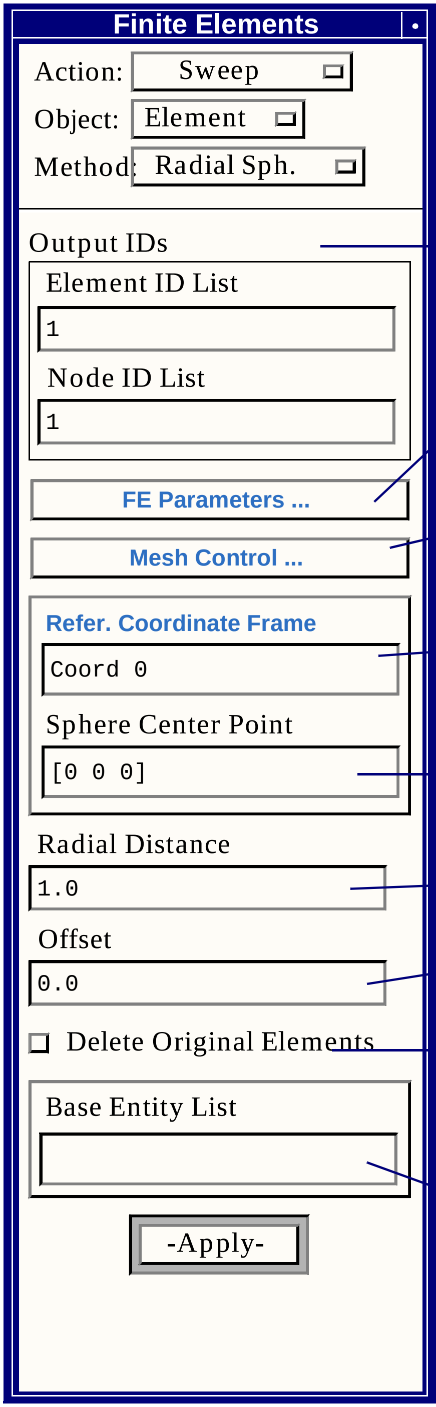
Distance to offset in radial direction from axis before starting sweep.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

The Radial Spherical Method

The Radial Spherical method allows creation of one or more elements by sweeping the base element through space radially outward from a center point.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially this form is raised because it contains required information.

Specifies the *Reference Coordinate Frame* in which the Sphere Center Point is defined. The default ID is the same as the active coordinate frame.

Specifies the sphere center point. All elements are swept out from this center point.

Total distance to sweep out from sphere center point.

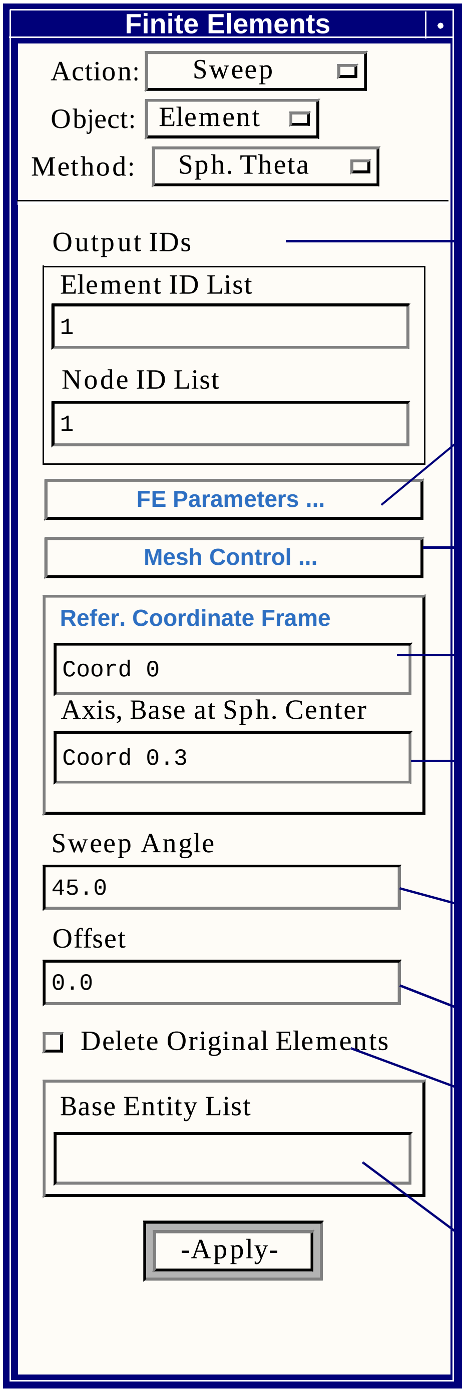
Offset distance in radial direction to beginning of sweep.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

The Spherical Theta Method

The Spherical Theta method allows creation of one or more elements by sweeping the base element through space along a path on a sphere that is like sweeping in the latitude direction in the earth's latitude and longitude system.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Specifies the *Reference Coordinate Frame* in which the Sphere Center Point is defined. The default ID is the same as the active coordinate frame.

Specifies the spherical center point and a direction to the "north pole" of the sphere. All elements are swept in the spherical theta direction with respect to this axis.

Specifies the angle, in degrees, to sweep the elements, with the positive direction being from the "north" pole towards the "equator" of the spherical system.

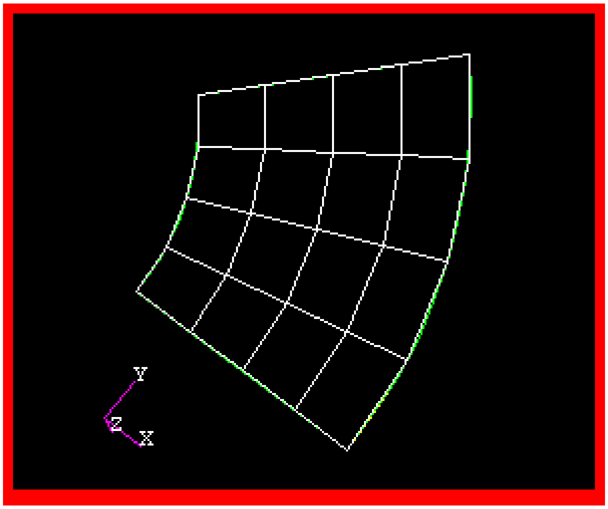
Specifies the offset angle, in degrees, to offset before beginning sweep.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

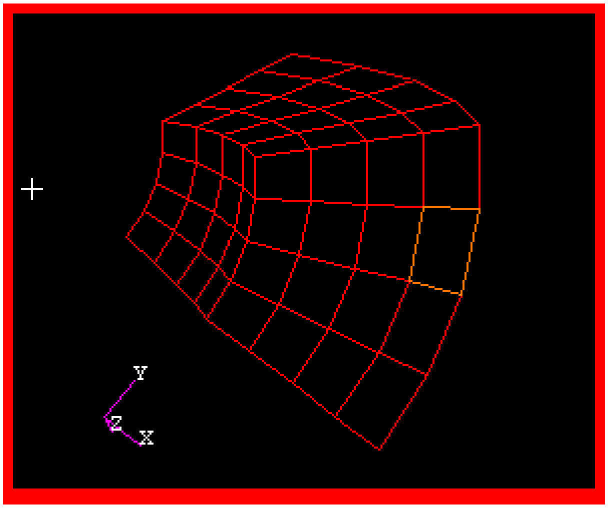
The following is an example of how the spherical theta method can be used to mesh a section of a hollow sphere:

Before:



The base entities are first set up. In this case, they are quads on a patch that is in the x-y plane. The default axis was used for this sweep (i.e., [0 0 0][0 0 1]). Therefore, the reference sphere for the sweeping process is one that is centered at the origin and has its pole along the z-axis.

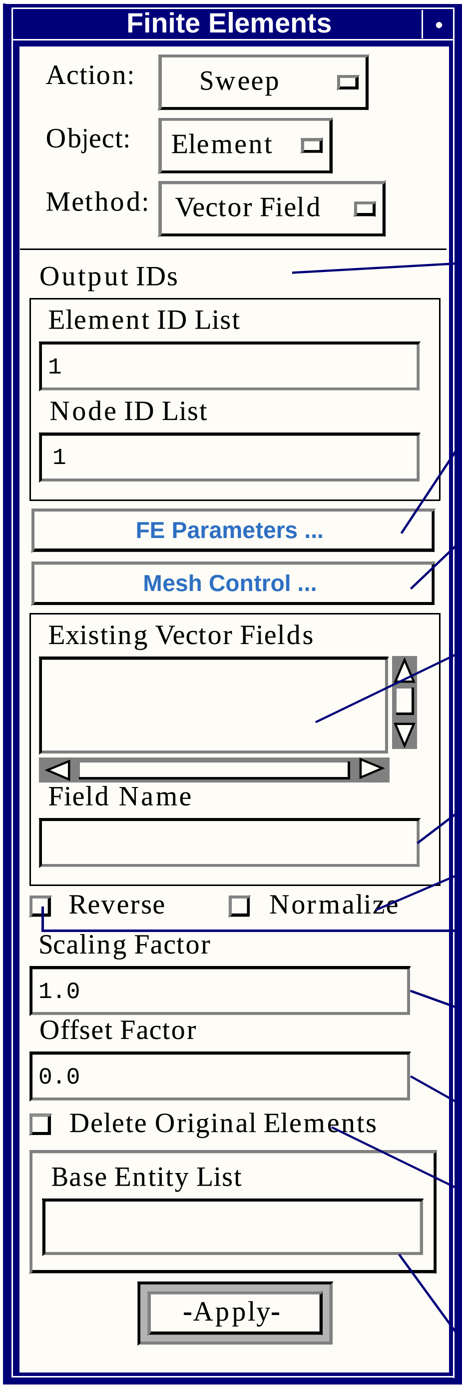
After:



The sweep direction is from the pole (in this case, the z-axis) toward the south pole, thereby creating a section of a hollow sphere.

The Vector Field Method

The Vector Field method allows creation of one or more elements by sweeping a base element in a direction determined by evaluating a vector field at each of its nodes.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially, this form is raised because it contains required information.

Form automatically lists all of the valid fields in the MSC.Patran database. Sweep only works for spatial vector fields that are created in a real (as opposed to parametric) coordinate system.

Specifies name of selected vector field.

Toggle ON to normalize the field vectors before sweeping.

Toggle ON to cause the sweep to occur in the reverse of the field direction.

Scaling factor to apply to magnitude of each field vector.

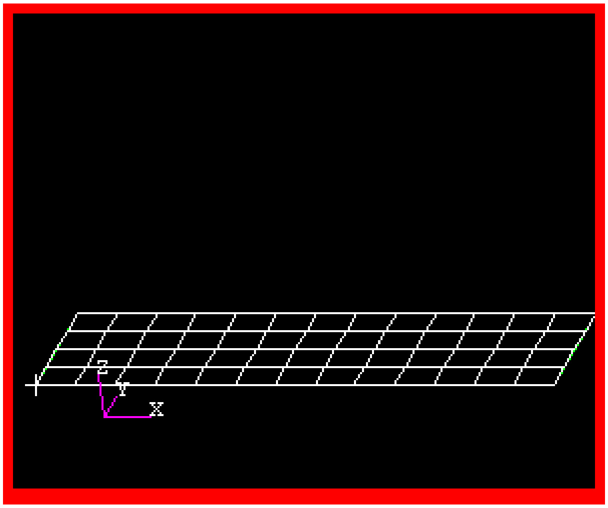
Factor multiplied by each vector to determine offset from each node in the base entity.

Toggle ON to delete base elements after the sweep. Nodes which are not used by other elements are also deleted. Element faces and edges, which may be in the base entity list, are not affected by this toggle.

Specifies a list of entities which are to be swept.

The following is an example of how the vector field sweep could be used:

Before:



The base entities are quads in the x-y plane. The underlying patch measures 2*PI units in the x direction and 1 unit in the y direction. The following vector field is used:

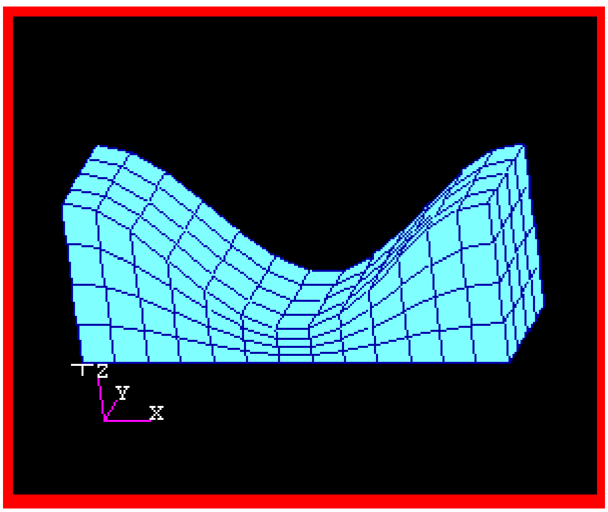
X = 0.0

Y = 0.0

Z = 1.5 + COS(X)

This field is evaluated at each of the *base* nodes to determine both the sweep direction and total sweep distance.

After:

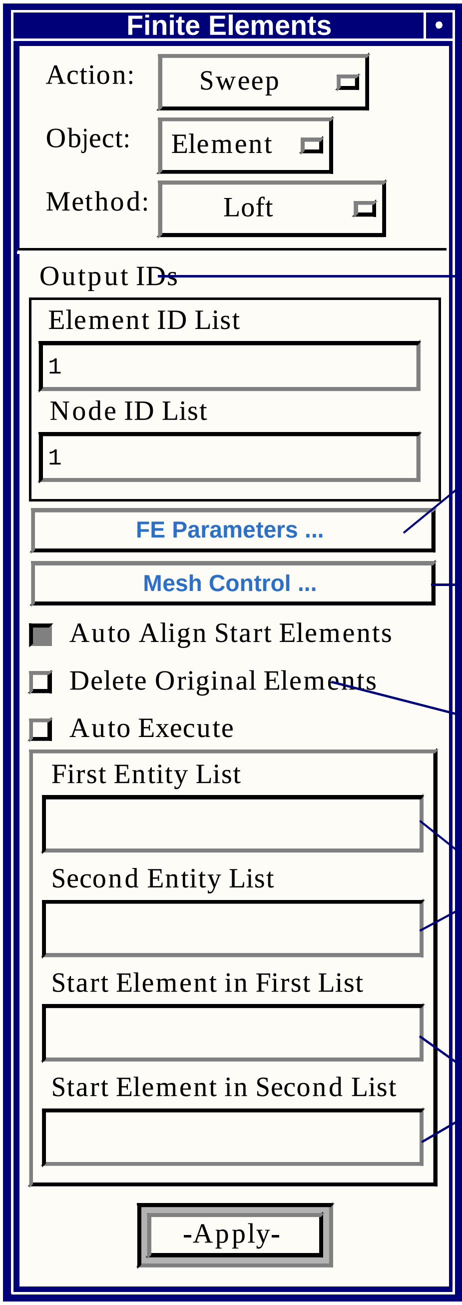


The mesh control is set for this example to create 4 equally sized elements in the direction of the sweep. The resulting mesh represents a cosine wave.

Although an equation was used to create the field (see **Spatial Field** (p. 140) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments*), other means can be used, such as filling in a table of data.

The Loft Method

The Loft method allows creation of one or more elements by sweeping a 2D base element to the location of a 2D top element. The two meshes have to be topological congruent.



Specifies **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for nodes and elements to be created.

Brings up the FE Parameters form. Use to define optional parameters for the elements to be swept. Normally, the default settings do not need to be modified.

Brings up the Mesh Control form. Use to define mesh control parameters. Initially this form is raised because it contains required information.

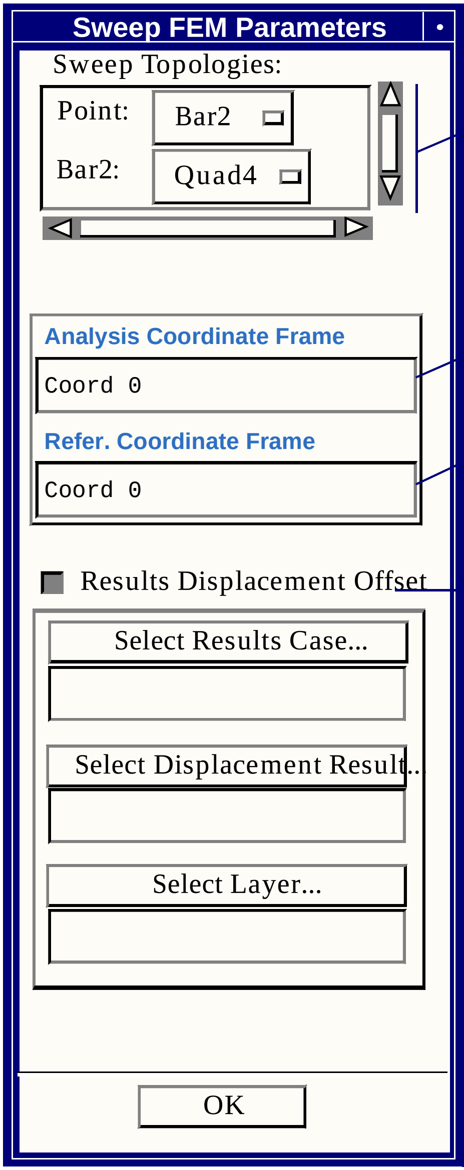
Toggle ON to delete base and top elements after the sweep. Nodes which are not used by other elements are also deleted.

Specifies a list of 2D elements which are to be swept. The first and second entity list defines two meshes, which will be connected with 3D elements. The number of elements in these two boxes has to be the same.

Specify a start element in first and second list. These two elements will be the first to be connected. If no elements are specified the first element in the entity lists will be used as a start element.

FEM Data

This form appears when the FE Parameters button is selected on any of the Sweep forms.



Specifies the type of element to be swept from each base element type listed.

Specifies the Analysis Coordinate Frame for all of the swept nodes.

Specifies the Reference Coordinate Frame for all of the swept nodes.

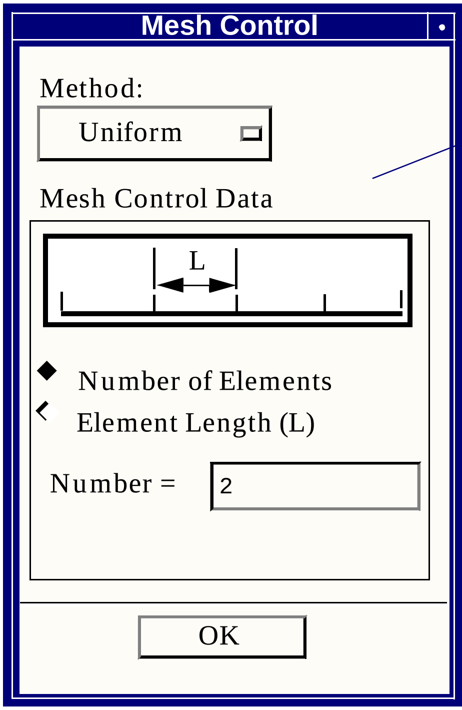
This feature, when turned **ON**, allows you to sweep elements based on rezoned nodal/element locations from displacement results of a previously run analysis.

For example, say you run an axisymmetric analysis of a wheel. From the displaced results of the axisymmetric model you want to create a full 3D model of the wheel to run subsequent analyses but from the deformed state of the axisymmetric analysis. To do this, you would **Select** the **Results Case...** of the axisymmetric model and then **Select** the **Displacement Result...** from which you want to create the rezoned mesh. (If multiple layers of results exist, you will have to **Select** the appropriate **Layer...** also.)

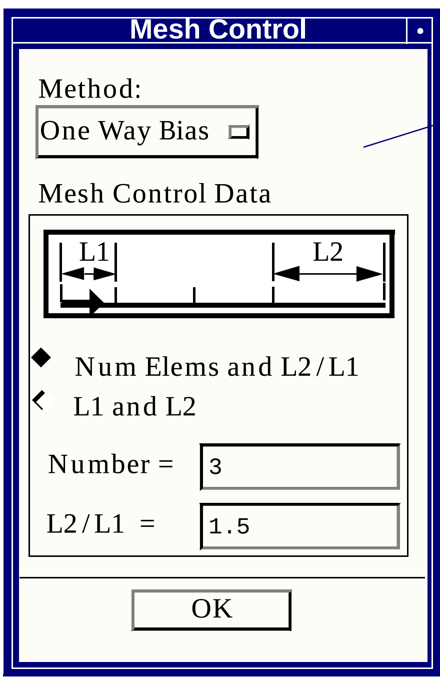
Naturally the results must exist in the database to perform this rezoned sweep. All sweep commands can use rezoning from displacement results except **Method** = **Loft.**

Mesh Control Data

Several Methods for defining either uniform or nonuniform discretization in the sweep direction are available. For the nonuniform methods, MSC.Patran will calculate the node spacing through a geometric progression based on the given L2/L1 ratio.



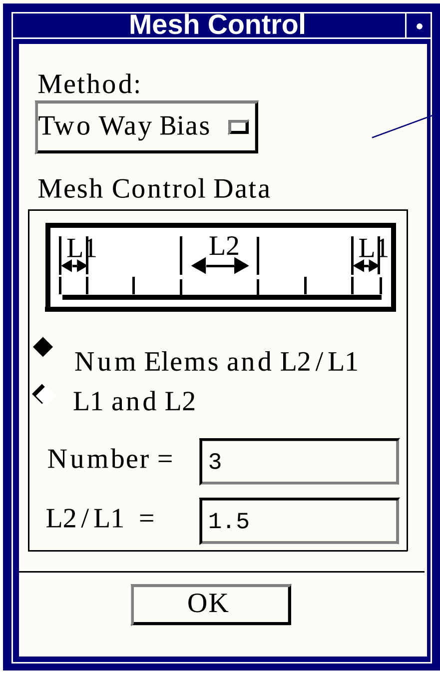
Defines a uniform discretization by selecting either “Number of Elements” or “Element Length(L).” If “Number of Elements” is selected, enter an integer value for the desired number of elements. If Element Length is selected, enter an element edge length (MSC.Patran will calculate the resulting number of elements needed, rounded off to the nearest integer value).



Defines a nonuniform discretization by selecting either “Number Elements and L2/L1” or L1 and L2.”

If “Number Elements and L2/L1” is selected, enter an integer value for the desired number of elements and an edge length ratio as indicated by the diagram. If “L1 and L2” is selected, enter edge lengths for the first and last elements.

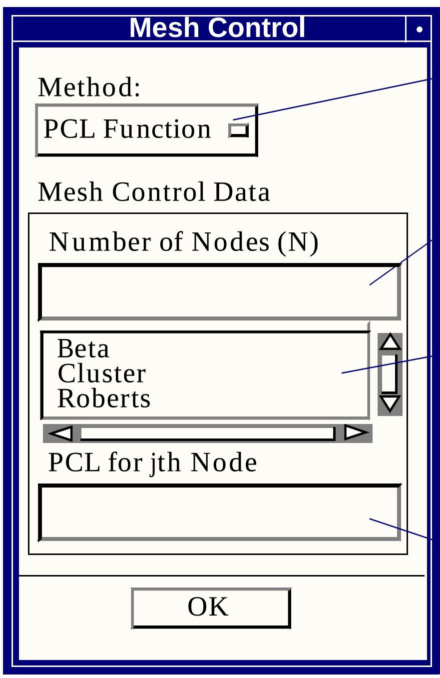
MSC.Patran will calculate the nonuniform node spacing through a geometric progression based on the given L2/L1 ratio. The direction arrow in the diagram indicates the sweep direction.



Defines a nonuniform discretization by selecting either “Number Elements and L2⁄L1” or L1 and L2.”

If “Number Elements and L2⁄L1” is selected, enter an integer value for the desired number of elements and an edge length ratio as indicated by the diagram. If “L1 and L2” is selected, enter edge lengths for the end and middle elements.

MSC.Patran will calculate the nonuniform node spacing through a geometric progression based on the given L2⁄L1 ratio. The direction arrow in the diagram indicates the sweep direction.



Defines an arbitrary set of node locations by either selecting a Predefined Function or by specifying a user-defined PCL function

Enter the number of nodes in the sweep direction.

Selecting a Predefined Function will enter its call into the “PCL for jth Node” data box.

To have a user-defined PCL function provide parametric node locations, enter its call(i.e.; my_function(j, N, 0.25)).

CHAPTER

6 The Renumber Action

- Introduction
- Renumber Forms

6.1 Introduction

Most often, ID numbers (IDs) for finite element nodes and elements are chosen and assigned automatically. The Renumber Action permits the IDs of nodes and elements to be changed. This capability is useful to:

- Offset the IDs of a specific list of entities.
- Renumber the IDs of all existing entities within a specified range.
- Compact the IDs of an entity type sequentially from 1 to N.

IDs must be positive integers. Duplicate IDs are not permitted in the List of New IDs, or in the selected Entity List (old IDs). A Starting ID or a List of New IDs may be entered in the input databox. If a finite element entity outside the list of entities being renumbered is using the new ID, the renumber process will abort since each entity must have a unique ID. The default is to renumber all the existing entities beginning with the minimum ID through the maximum ID consecutively starting with 1.

If only one ID is entered, it is assumed to be the starting ID. The entities will be renumbered consecutively beginning with the starting ID.

If more than one ID is entered, then there must be at least as many new IDs as there are valid old IDs. If there are fewer IDs in the List of New IDs than there are valid IDs in the selected Entity List, renumbering will not take place and a message will appear in the command line indicating exactly how many IDs are needed. The List of New IDs may not contain a #. However, the list may have more IDs than needed.

Important: Try to estimate the number of IDs needed. A large number of unnecessary IDs will slow down the renumber process.

The IDs in the selected Entity List may contain a #. The value of the maximum existing ID is automatically substituted for the #. There may be gaps of nonexisting entities in the list but there must be at least one valid entity ID in order for renumbering to take place.

A percent complete form shows the status of the renumber process. When renumbering is complete, a report appears in the command line indicating the number of entities renumbered and their new IDs. The renumber process may be halted at any time by pressing the Abort button and the old IDs will be restored.

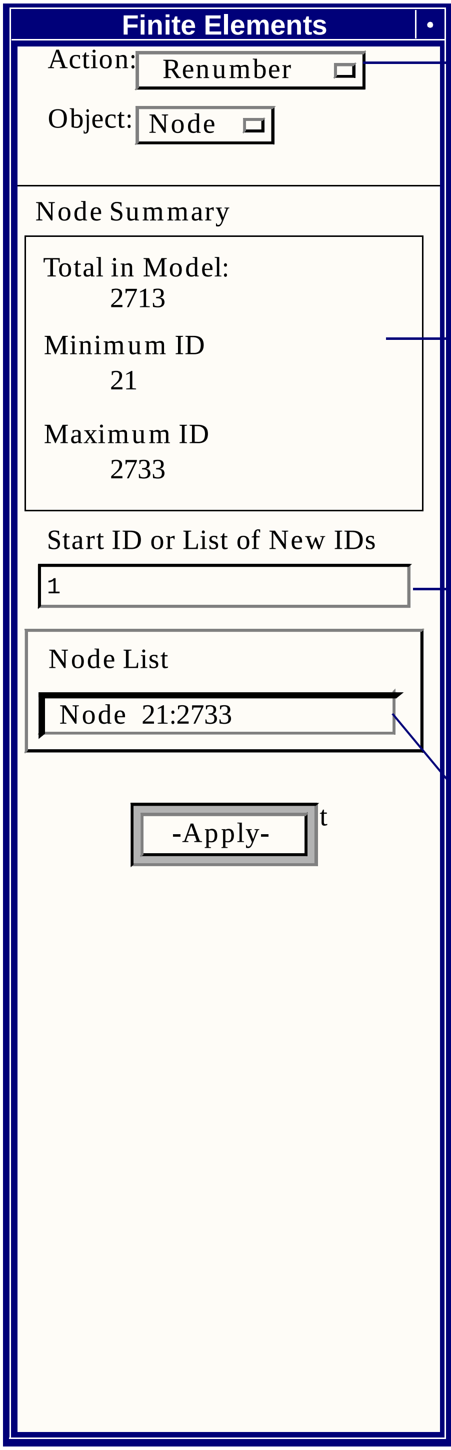
6.2 Renumber Forms

When *Renumber* is the selected Action the following options are available.

Object	Description
Node	The node menu selection provides the capability to renumber or change the IDS of nodes.
Element	The element menu selection provides the capability to renumber or change the IDs of elements.
MPCs	The MPC menu selection provides the capability to renumber or change the IDS of MPCs.

Renumber Nodes

Figure 6-1



Use this option to renumber nodes. Each node has a unique node ID. See **Introduction** (p. 156).

Shows how many nodes exist in the model and minimum/maximum values of node IDs. **Note:** All nodes are numbered sequentially when the Maximum ID is equal to Total in Model.

Specifies the starting ID, or a list of *new* node IDs to assign. Node IDs must be positive integers. Although # is not a valid entry here, a large number may be entered. If the number of IDs is less than the number of valid nodes, renumbering will not take place.

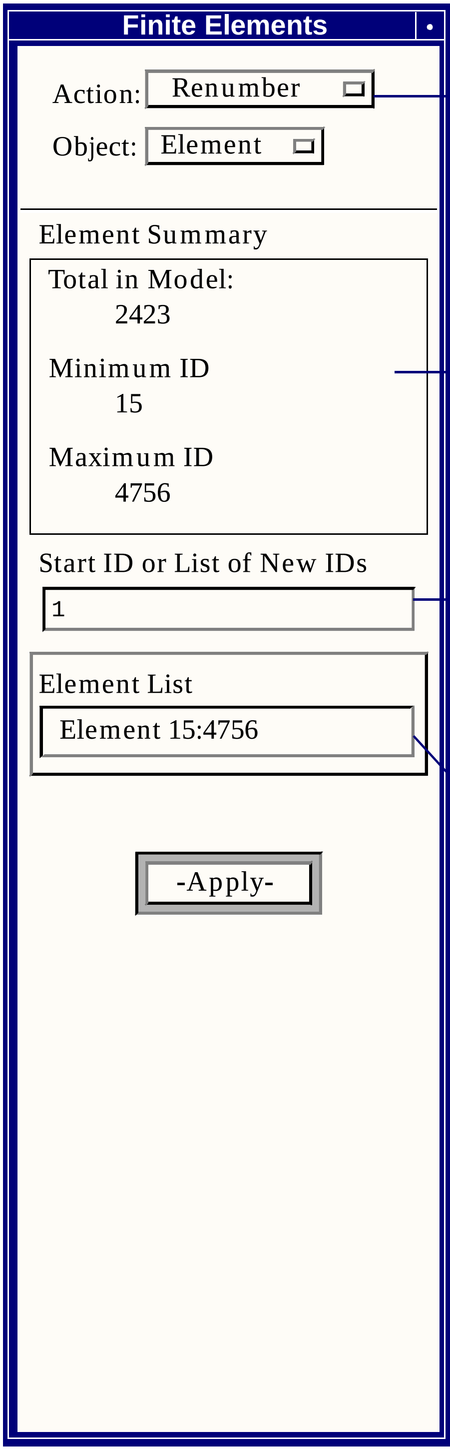
Specifies which *old* nodes are to be renumbered. A list of nodes can be entered here or an active group of nodes can be selected from the viewport.

The default is to renumber all nodes (Node minimum ID to maximum ID) consecutively beginning with the Start ID. The entry, Node 1:#, is also valid to indicate all nodes. There may be gaps of nonexisting nodes in the list, but there must be at least one valid node in order for renumbering to take place. Duplicate IDs are not permitted.

If a node outside the list of nodes being renumbered is using the new ID, the renumber process will abort since each node must have a unique ID.

Renumber Elements

Figure 6-2



Use this option to renumber elements. Each element has a unique element ID. See **Introduction** (p. 156).

Shows how many elements exist in the model and minimum/maximum values of element IDs. **Note:** Elements are numbered sequentially when the Maximum ID is equal to Total in Model.

Specifies the starting ID or list of *new* element IDs to assign. Element IDs must be positive integers. Although # is not a valid entry here, more IDs than needed may be entered. If there are too few IDs or if there are duplicate IDs in the list, renumbering will not take place.

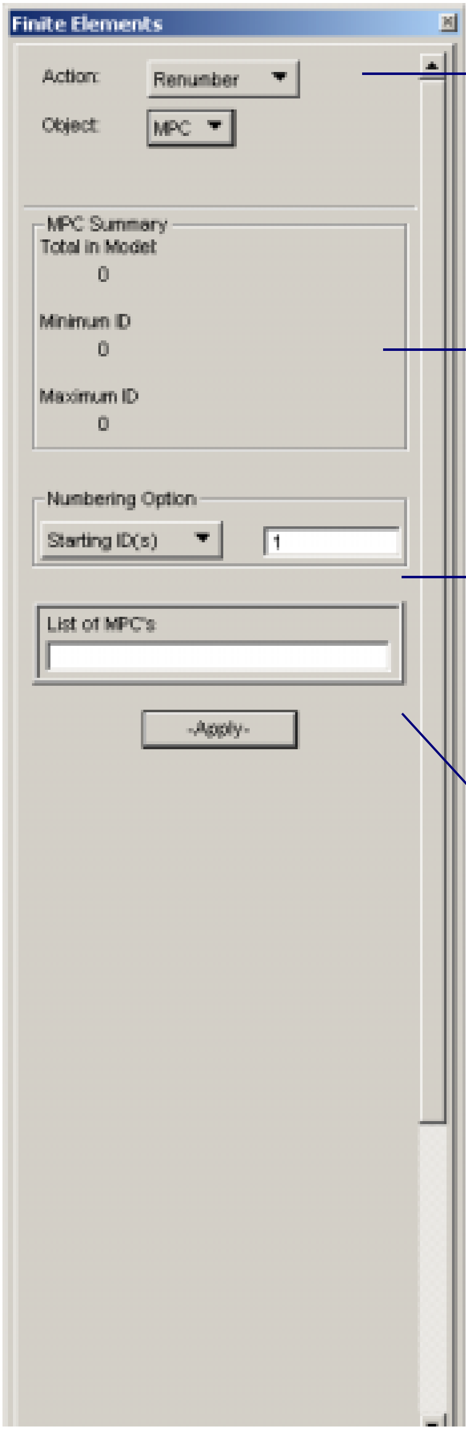
Specifies which *old* elements are to be renumbered. A list of elements can be entered here or an active group of elements can be selected from the viewport.

The default is to renumber all elements (Element minimum ID to maximum ID) consecutively beginning with the Start ID. The entry, Element 1:#, is also valid to indicate all elements.There may be gaps of nonexisting elements in the list, but there must be at least one valid element in order for renumbering to take place. Duplicate IDs are not permitted.

If an element outside the list of elements being renumbered is using the new ID, the renumber process will abort since each element must have a unique ID.

Renumber MPCs

Figure 6-3



Use this option to renumber mpcs. Each mpc has a unique ID. See **Introduction** (p. 156).

Shows how many MPCs exist in the model and minimum/maximum values of MPC IDs. **Note:** MPCs are numbered sequentially when the Maximum ID is equal to Total in Model.

Specifies the starting ID or list of *new* MPC IDs to assign. MPC IDs must be positive integers. Although # is not a valid entry here, more IDs than needed may be entered. If there are too few IDs or if there are duplicate IDs in the list, renumbering will not take place. MSC.Patran by default does not number elements and MPCs with the same ID.

Specifies which *old* MPCs are to be renumbered. A list of MPCs can be entered here or an active group of MPCs can be selected from the viewport.

The default is to renumber all MPCs (MPC minimum ID to maximum ID) consecutively beginning with the Start ID. The entry, MPC 1:#, is also valid to indicate all MPCs.There may be gaps of nonexisting MPCs in the list, but there must be at least one valid MPC in order for renumbering to take place. Duplicate IDs are not permitted.

If an MPC outside the list of MPCs being renumbered is using the new ID, the renumber process will abort since each MPC must have a unique ID. Warnings are given for element and MPC ID conflicts.

CHAPTER 7

The Associate Action

- Introduction
- Associate Forms

7.1 Introduction

The purpose of the Associate Action is to define a logical connection between geometry and finite elements. The associate action allows users to associate finite element entities to geometries, if they are unassociated, thereby enabling the user to apply loads, boundary conditions and properties directly to the geometry instead of to the individual finite element entities. When associating finite elements to geometric entities, two general rules apply:

Rule 1:The nodes are associated with the lowest order existing topological entity first which is a vertex, then an edge, face, and body.

Rule 2:The finite elements are associated with the same order geometric entity, i.e., a beam element with a curve, or a quad element with a surface.

A typical application would be the importing of an IGES file which has both a geometry and a finite element model. However, there is no associativity between either of the models. The Associate Action will provide the capability of logically connecting the two models together, thus defining an associativity between them.

Association of elements and nodes are based on their geometric proximity to the selected geometry. When associating elements to geometry (except points) users have the option of specifying whether or not a “mesh definition” must be created on the curves or edges. This option creates an implicit mesh record on the curve that allows the mesher to create congruent meshes across neighboring geometries.

CAUTION: When a mesh is associated, to say a surface, and “mesh definition” is requested to be created, if a “mesh definition” already exists on an edge of the surface a warning is issued about a possible non congruent mesh along that edge. This is because the associate code simply duplicates the existing mesh definition as multiple mesh definitions cannot exist on an edge to produce a congruent mesh.

Four methods for associating nodes and finite elements to geometry are provided: Point, Curve, Surface, and Solid.

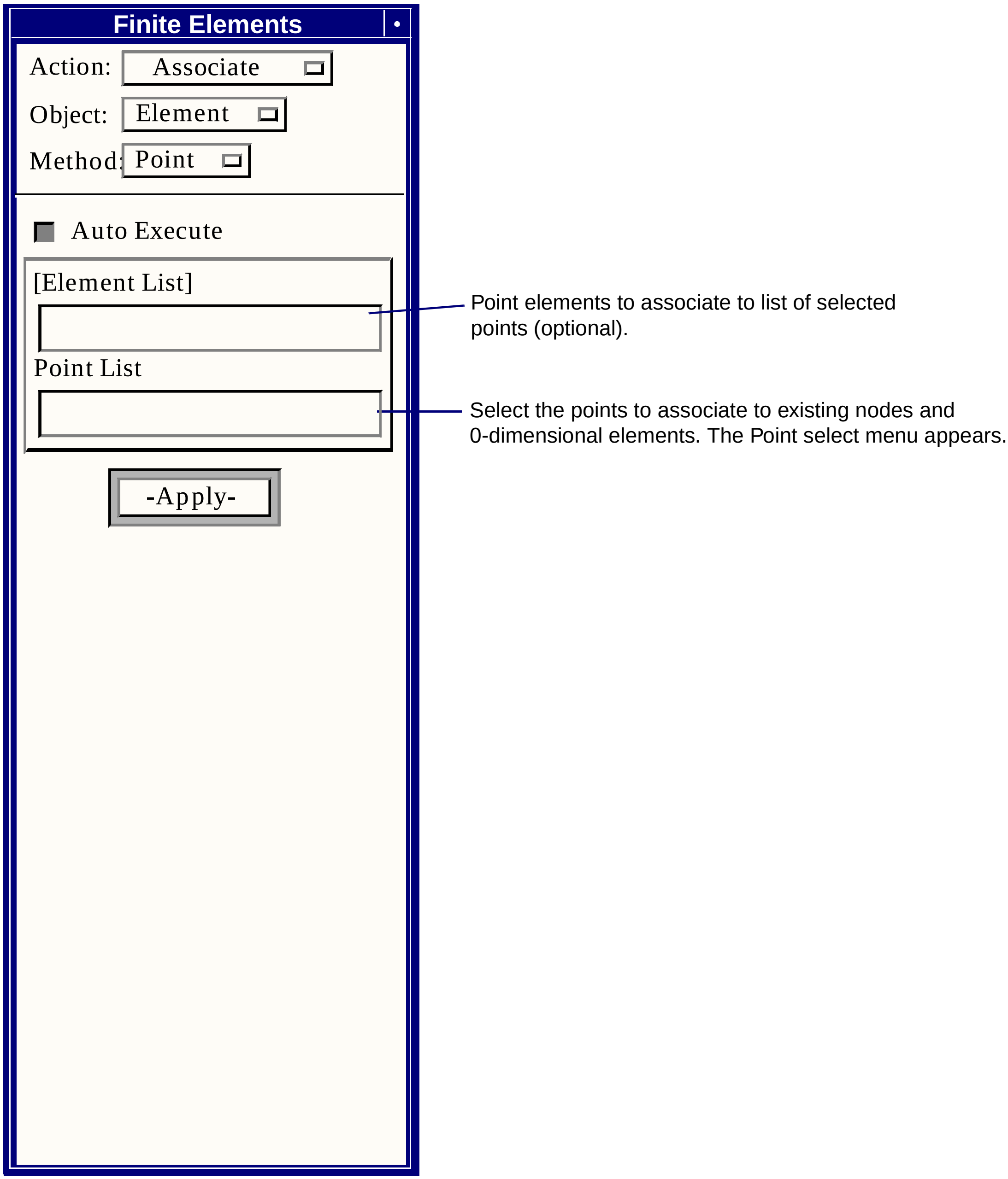
7.2 Associate Forms

The following options are available when *Associate* is the selected Action and *Element* is the selected Object.

Method	Description
Point	The Point method allows the association of nodes and 0-dimensional finite elements to geometric point entities.
Curve	The Curve method allows the association of nodes and 1-dimensional finite elements to topological vertices and edges and geometric curves respectively.
Surface	The Surface method allows the association of nodes and 2-dimensional finite elements to topological vertices, edges, and faces and geometric surfaces respectively.
Solid	The Solid method allows the association of nodes and 3-dimensional finite elements to topological vertices, edges, faces, and bodies and geometric solids respectively.

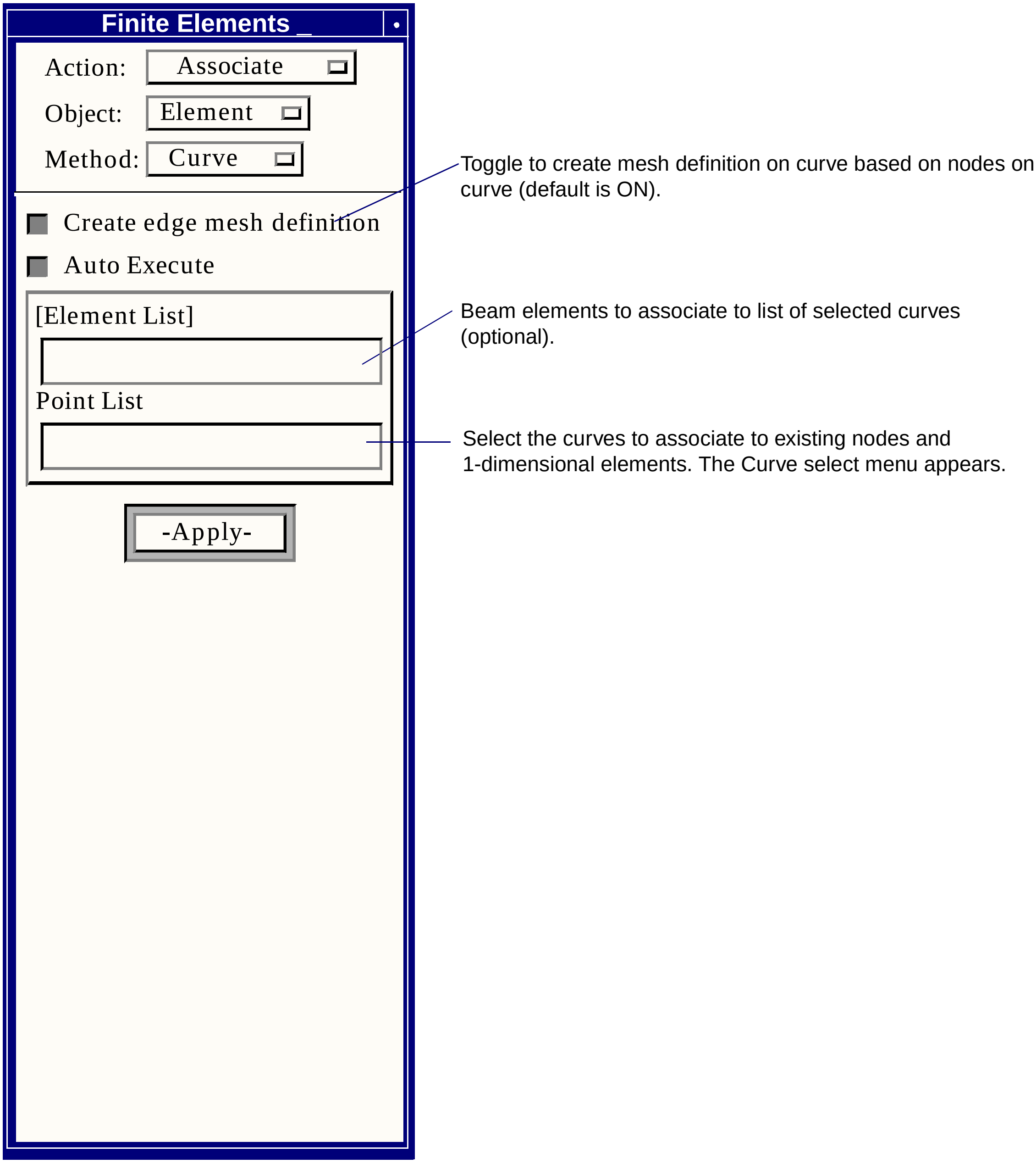
The Point Method

The Point method allows the association of nodes and 0-dimensional finite elements to geometric point entities. The associate action allows users to associate finite element entities to geometries, if they are unassociated, thereby enabling the user to apply loads, boundary conditions and properties directly to the geometry instead of to the individual finite element entities.



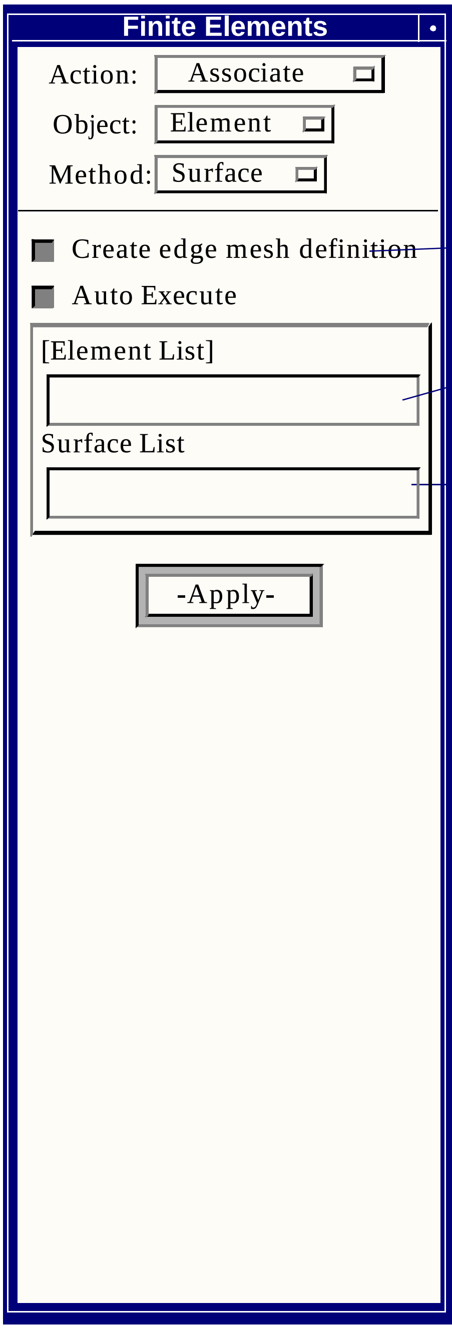
The Curve Method

The Curve method allows the association of nodes and 1-dimensional finite elements to geometric curve entities. The associate action allows users to associate finite element entities to geometries, if they are unassociated, thereby enabling the user to apply loads, boundary conditions and properties directly to the geometry instead of to the individual finite element entities.



The Surface Method

The Surface method allows the association of nodes and 2-dimensional finite elements to geometric surface entities. The associate action allows users to associate finite element entities to geometries, if they are unassociated, thereby enabling the user to apply loads, boundary conditions and properties directly to the geometry instead of to the individual finite element entities.



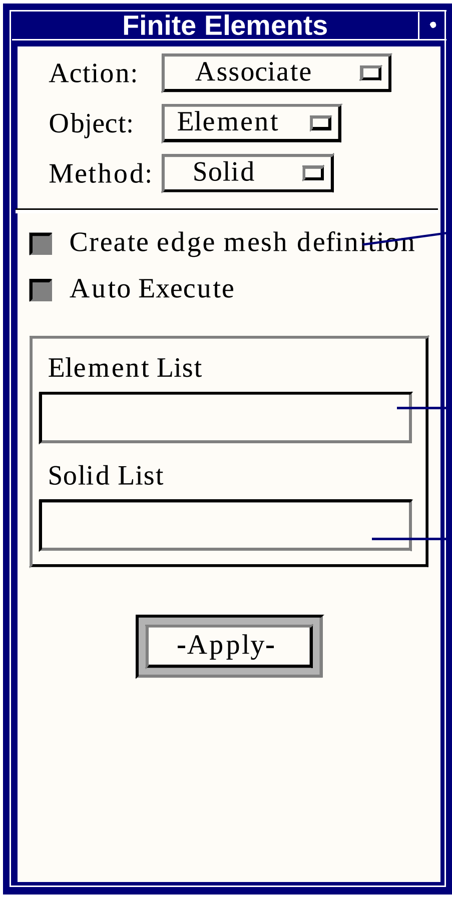
Toggle to create mesh definition on surface edge based on nodes on edge (default is ON).

Surface elements to associate to list of selected surfaces (optional).

Select the surfaces to associate to existing nodes and 2-dimensional elements. The Surface select menu appears.

The Solid Method

The Solid method allows the association of nodes and 3-dimensional finite elements to geometric solid entities. The associate action allows users to associate finite element entities to geometries, if they are unassociated, thereby enabling the user to apply loads, boundary conditions and properties directly to the geometry instead of to the individual finite element entities.



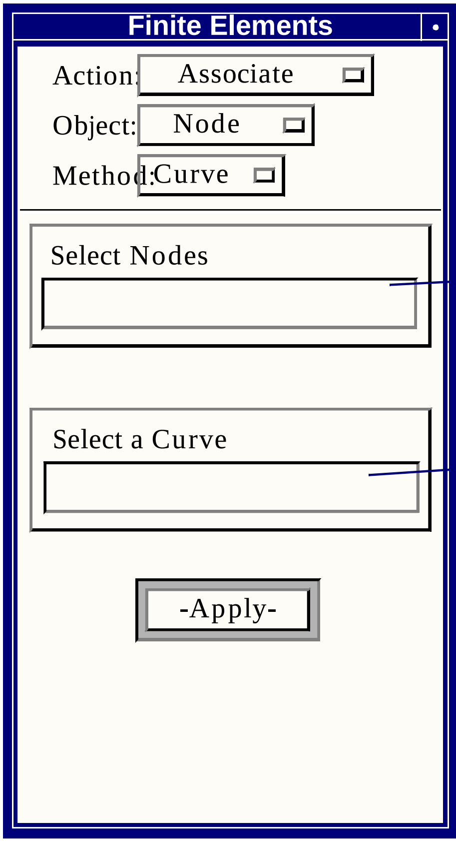
Toggle to create mesh definition on solid edge based on nodes on edge (default is ON).

Select the 3-dimensional elements to associate to existing solid entities in the Solid List. The Solid Element select menu appears.

Select the solids to associate to existing nodes and 3-dimensional elements in the Element List. The Solid select menu appears.

The Node Forms

This form is used to associate nodes and curves.



Select the nodes that will be associated with the curve.

Select the curve that will be associated with the nodes.

CHAPTER

8 The Disassociate Action

- Introduction
- Disassociate Forms

8.1 Introduction

The Finite Element Disassociate action allows the user to disassociate a finite element entity (a node or an element) either by its geometric association or by ID. When a geometry is selected for disassociation, all finite element entities of the selected type associated to that geometry get disassociated. When an ID is selected, only the selected item is disassociated.

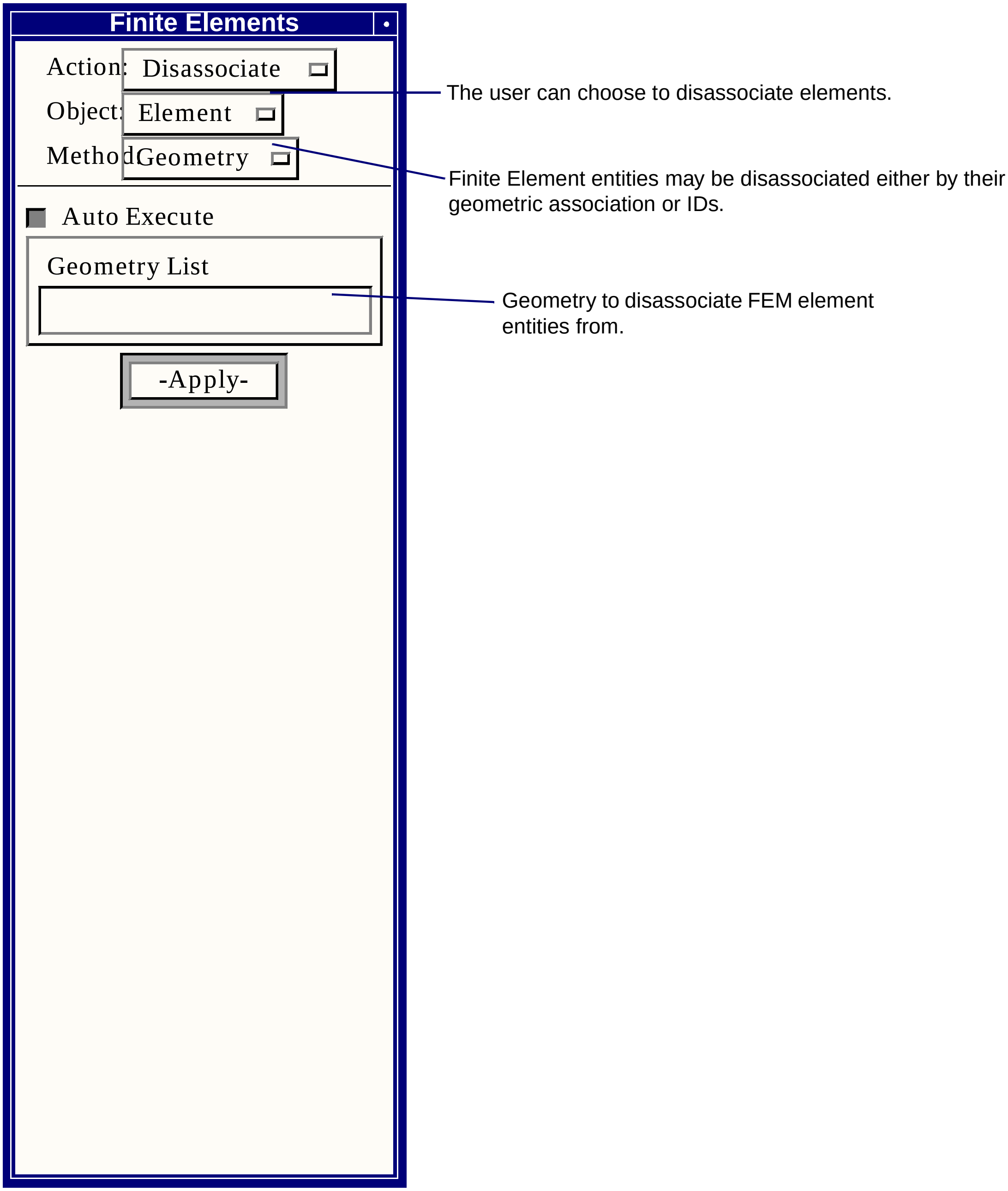
8.2 Disassociate Forms

The following table shows the possible methods by which Finite Element entities could be disassociated.

Method	Description
Elements	Disassociate elements associated to the picked geometry.
	Disassociate elements with specified IDs from their parent geometry.
Node	Disassociate nodes associated to the picked geometry.
	Disassociate nodes with specified IDs from their parent geometry.

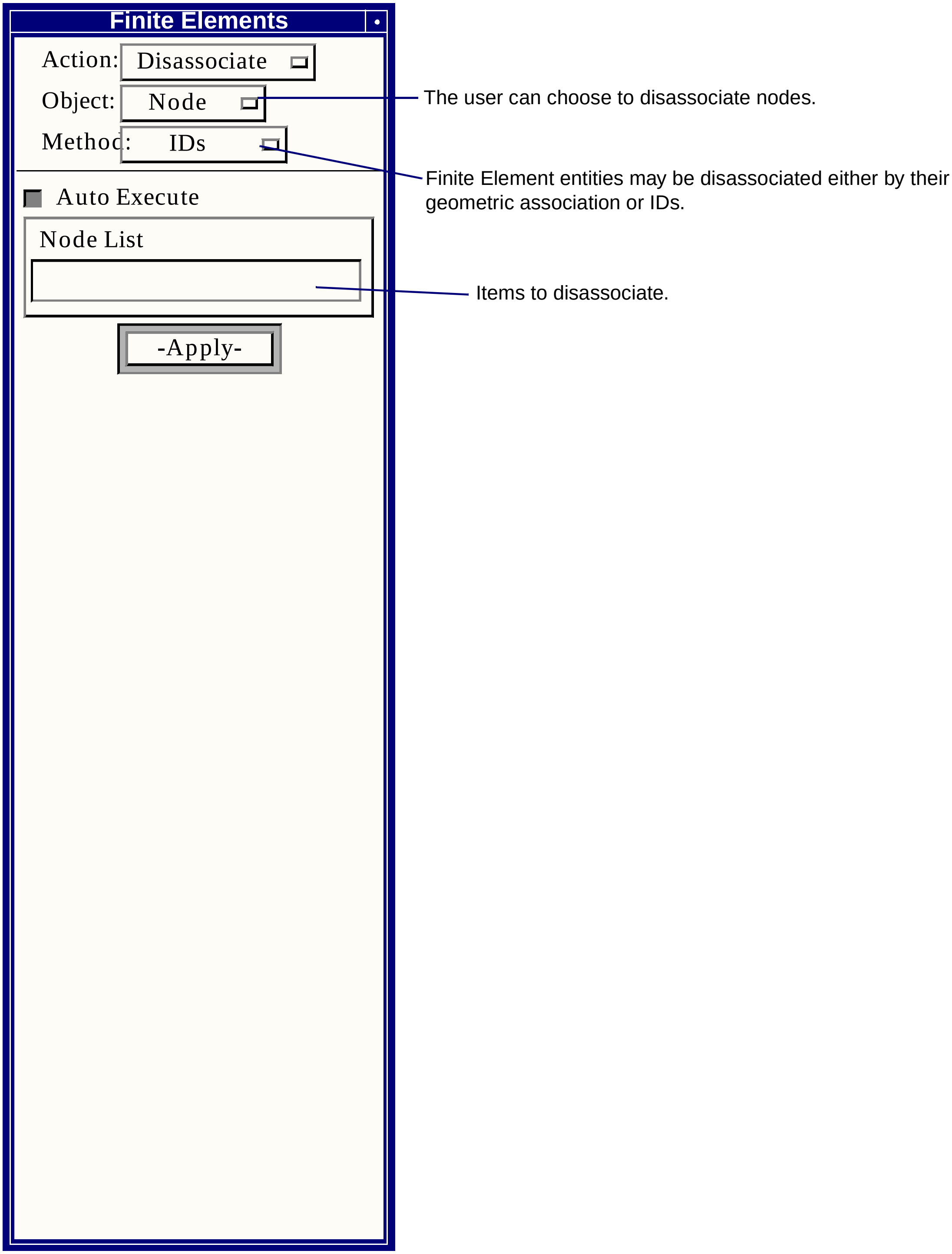
Elements

The elements may be disassociated from their parent geometry either by picking the parent geometry, in which case all the Finite elememt entities of the chosen type associated to the parent geometry will get disassociated, or by picking individual IDs.



Node

The nodes may be disassociated from the parent geometry either by picking the parent geometry, in which case all the FEM entities of the chosen type associated to the picked geometry will be disassociated, or by picking the individual IDs.



CHAPTER

9 The Equivalence Action

- Introduction to Equivalencing
- Equivalence Forms

9.1 Introduction to Equivalencing

Equivalencing is the process of reducing all nodes that coexist at a point to a single node. This change is propagated through any existing FEM definition (element connectivity definitions, MPC equations, loads and boundary conditions), geometry definition and groups.

By default, a red highlight circle is drawn over each retained node causing the deletion of neighboring nodes. For example, if nodes 2 and 3 are deleted because of their proximity to node 1, then a circle is drawn over node 1. If node labels are active, a highlight label appears indicating the selected ID.

The removal of a node by equivalencing causes all occurrences of that node in the FEM definition to be replaced with the surviving node, which is usually the coincident node with the lowest ID. The surviving node remains associated with whatever geometric entity it was associated with prior to equivalencing. However, the effect on groups are additive. For example, if equivalencing removes a node which belongs to group1, in favor of a surviving node which belongs to group2, then the surviving node is associated with both groups.

The selection of the retained node among a set of coincident nodes is guided by two principles:

1. The node with the lowest ID should be retained.
2. Equivalencing must never cause element edge collapse or the removal of an MPC equation or zero length element, such as a spring or mass.

Therefore, MSC.Patran always retains the coincident node with the lowest ID, unless one of the coincident nodes belongs to an MPC or a zero length element edge, and the MPC or element contains at least two nodes in the set of nodes for which equivalencing has been requested. (In the Equivalence-All option, for example, that set is the set of all nodes in the model.) Furthermore, if nodes 1, 2, and 3 are coincident and nodes 2 and 3 are connected by an MPC equation, then if the Equivalence-All option is chosen, all references to node 1 will be replaced with node 2. However, if the Equivalence-List option is used with a node list of “Node 1:2”, then all references to node 2 will be replaced with node 1. The MPC is ignored here because only one of its nodes is in the user-specified set.

The automated equivalencing method available in MSC.Patran is called Geometric Equivalencing. Geometric Equivalencing is based upon the physical coordinates of the node points. The proximity is compared with a user definable tolerance parameter called the Equivalencing Tolerance.

Equivalencing can be delayed until the completion of the model, but it is generally recommended that equivalencing be performed before loads and boundary conditions are defined. In this way, diagnostics which may be issued for loads and boundary conditions will have more significance since MSC.Patran will be implementing the values of nodal attributes at common nodes at the time of loads and boundary specification.

Equivalencing should always be performed prior to the optimization of element connectivity and the generation of the neutral file output file.

The model, or any portion of the model, can be equivalenced more than once. When the new component is completed and equivalenced, only those nodes which are newly equivalenced as a result of this second equivalencing will be circled.

It is necessary to perform local equivalencing whenever a modification is made to a region’s mesh. Only the new nodes will be subject to equivalencing.

If the INTERRUPT button is selected during equivalencing, the search for equivalent nodes is immediately terminated. If any changes have been made to the node numbering sequence, they will be reversed.

The results of equivalencing can be verified by bringing up the “Verify/Element/Boundaries” form.

9.2 Equivalence Forms

When Equivalence is the selected Action the following options are available.

Object	Method	Description
All	Tolerance Cube	Equivalence the whole model using tolerance cube.
	Tolerance Sphere	Equivalence the whole model using tolerance sphere.
Group	Tolerance Cube	Equivalence only nodes in groups specified using tolerance cube.
	Tolerance Sphere	Equivalence only nodes in groups specified using tolerance sphere.
List	Tolerance Cube	Equivalence nodes in user-defined lists by cube tolerance.
	Tolerance Sphere	Equivalence nodes in user-defined lists by sphere tolerance.

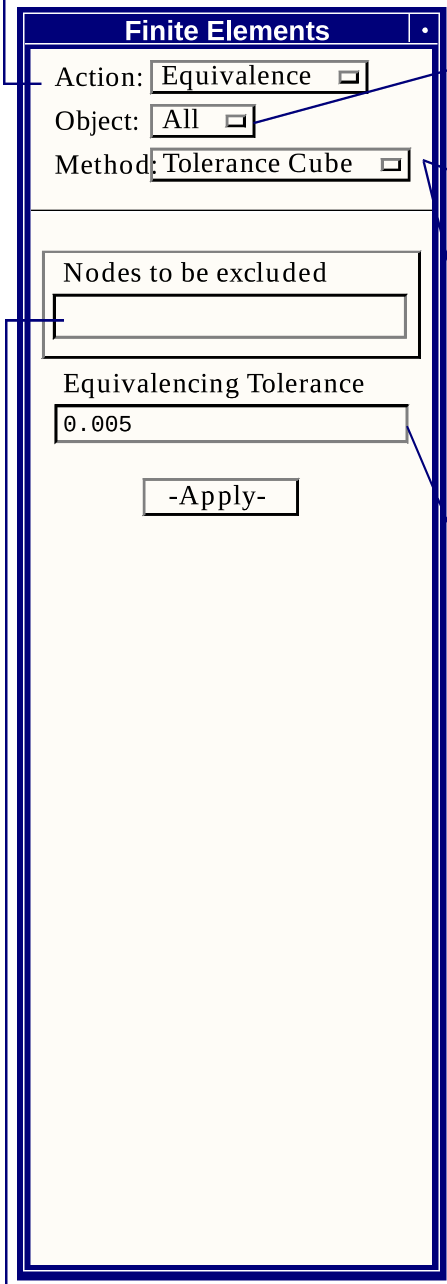
Equivalence - All

Note: You can now generate a Node Equivalence Report by setting the environment variable "WRITE_EQUIVALENCE_REPORT" to "YES". To set the variable, type:

```
setenv WRITE_EQUIVALENCE_REPORT , YES
```

in **The settings.pcl file** (p. 41).

For information on the Equivalence action, see **Introduction to Equivalencing** (p. 176).



Use this option to have all nodes in the model considered for equivalencing.

The following are methods available for equivalencing:

Tolerance Cube The equivalencing procedure which uses a cube and is the default method used for equivalencing. If Tolerance Cube is selected, then two node points are equivalenced if all of their coordinates in the global Cartesian frame lie within the tolerance of each other. The node with the lower ID is always retained.

Tolerance Sphere Uses a sphere for equivalencing. If Tolerance Sphere is chosen, two node points are equivalenced if the distance between them is within the tolerance.

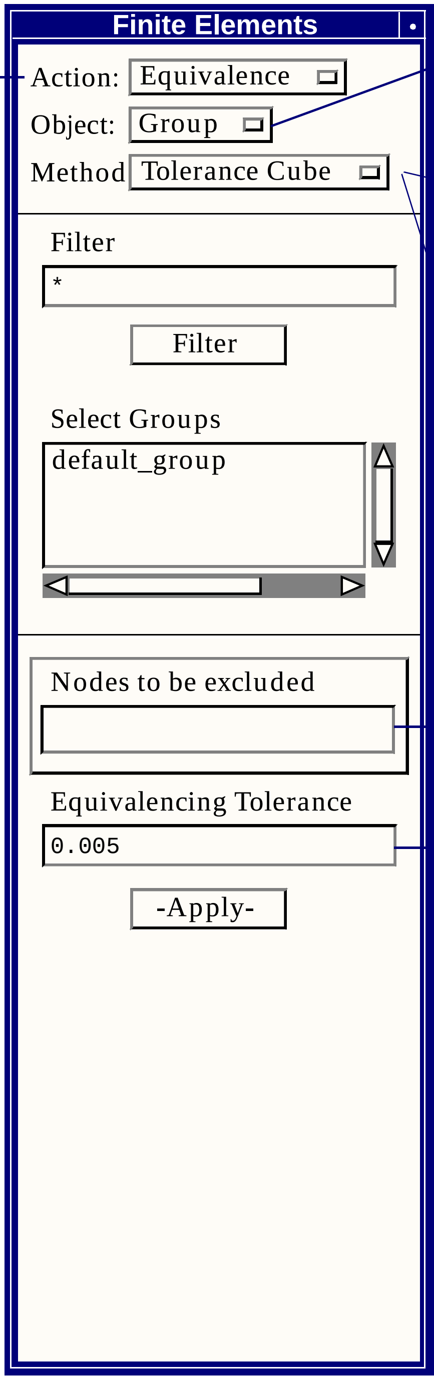
A user-definable tolerance parameter used to determine whether two nodes are sufficiently close to be considered coincident, and therefore are subject to nodal equivalencing. By default, this parameter is equal to the Global Model Tolerance set in Global Preferences. The value in the Equivalencing Tolerance databox is used for equivalencing, but the value set in Global Preferences will continue to appear every time equivalencing is reselected.

The tolerance should never be set too low (less than 10.E-7) since computational round-off can cause two otherwise identical points to be slightly offset. As part of the equivalencing computations, MSC.Patran internally calculates the minimum tolerance that will ensure that no element edges will collapse. If this calculated tolerance is less than the user selected tolerance, then the calculated tolerance is used and a message is issued.

All selected nodes will be excluded from equivalencing.

Equivalence - Group

For information on the Equivalence action, see **Introduction to Equivalencing** (p. 176).



Use this option to have only those nodes belonging to a particular set of groups considered for equivalencing.

The following are methods available for equivalencing:

Tolerance Cube — Uses a cube and is the default method used for equivalencing. If Tolerance Cube is selected, then two node points are equivalenced if all of their coordinates in the global Cartesian frame lie within the tolerance of each other. The node with the lower ID is always retained.

Tolerance Sphere — Uses a sphere for equivalencing. If Tolerance Sphere is chosen, two node points are equivalenced if the distance between them is within the tolerance.

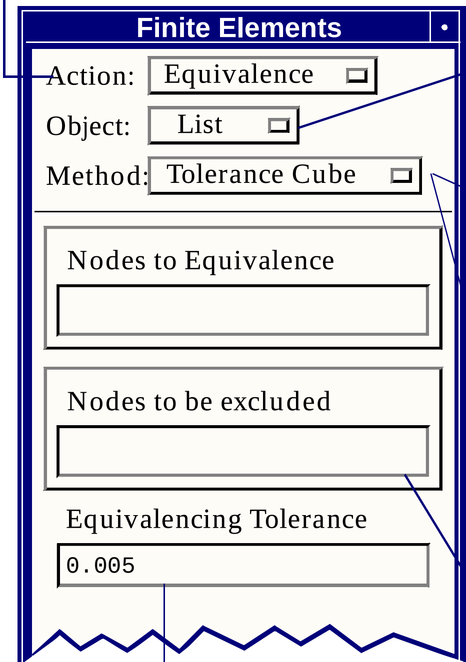
All selected nodes will be excluded from equivalencing.

A user-definable tolerance parameter used to determine whether two nodes are sufficiently close to be considered coincident and therefore are subject to nodal equivalencing. By default, this parameter is equal to the Global Model Tolerance set in Global Preferences. The value in the Equivalencing Tolerance databox is used for equivalencing, but the value set in Global Preferences will continue to appear every time equivalencing is reselected. The tolerance should never be set too low (less than 10.E-7) since computational round-off can cause two otherwise identical points to be slightly offset. As part of the equivalencing computations, MSC.Patran internally calculates the minimum tolerance that will ensure that no element edges will collapse. If this calculated tolerance is less than the user selected tolerance, then the calculated tolerance is used and a message is issued.

Equivalence - List

Figure 9-1

For information on the Equivalence action, see **Introduction to Equivalencing** (p. 176).



Use this option to have only nodes in a user- defined set considered for equivalencing.

The following are methods available for equivalencing:

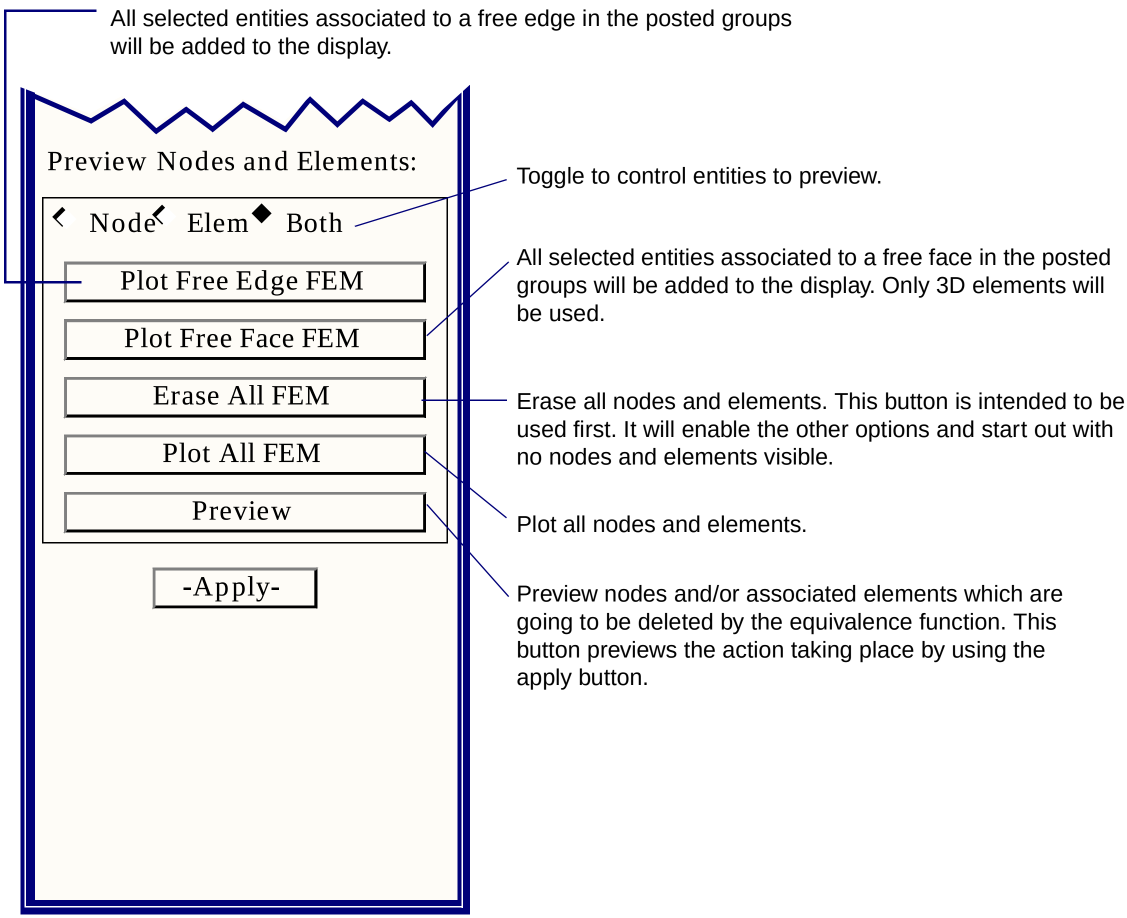
Tolerance Cube Uses a cube and is the default method used for equivalencing. If Tolerance Cube is selected, then two node points are equivalenced if all of their coordinates in the global Cartesian frame lie within the tolerance of each other. The node with the lower ID is always retained.

Toler-ance Sphere Uses a sphere for equivalencing. If Tolerance Sphere is chosen, two node points are equivalenced if the distance between them is within the tolerance.

All selected nodes will be excluded from equivalencing.

A user-definable tolerance parameter used to determine whether two nodes are sufficiently close to be considered coincident and therefore are subject to nodal equivalencing. By default, this parameter is equal to the Global Model Tolerance set in Global Preferences. The value in the Equivalencing Tolerance databox is used for equivalencing, but the value set in Global Preferences will continue to appear every time equivalencing is reselected.

The *tolerance should never be set too low* (less than 10.E-7) since computational round-off can cause two otherwise identical points to be slightly offset. As part of the equivalencing computations, MSC.Patran internally calculates the minimum tolerance that will ensure that no element edges will collapse. If this calculated tolerance is less than the user selected tolerance, then the calculated tolerance is used and a message is issued.


All selected entities associated to a free edge in the posted groups will be added to the display.
Preview Nodes and Elements:
Node
Elem
Both
Toggle to control entities to preview.
Plot Free Edge FEM
Plot Free Face FEM
All selected entities associated to a free face in the posted groups will be added to the display. Only 3D elements will be used.
Erase All FEM
Erase all nodes and elements. This button is intended to be used first. It will enable the other options and start out with no nodes and elements visible.
Plot All FEM
Plot all nodes and elements.
Preview
Preview nodes and/or associated elements which are going to be deleted by the equivalence function. This button previews the action taking place by using the apply button.
-Apply-

CHAPTER

10

The Optimize Action

- Introduction to Optimization
- Optimizing Nodes and Elements
- Selecting an Optimization Method

10.1 Introduction to Optimization

The purpose of optimization is to renumber the nodes or elements of a model in such a way that the stiffness matrix assembled in a finite element analysis can be solved (inverted) by using a minimum of CPU time, memory, and disk space.

The solvers, used by finite element codes, take advantage of the fact that the stiffness matrix is symmetric, banded, and sparse (see **Figure 10-1**). The cost (CPU time, memory, and disk space) of solving the matrix is determined by the *sparsity* or zero-nonzero characteristics of the matrix. The sparsity is affected by the numbering of the nodes, or elements, depending on the solver. In general, the attributes of the matrix (see **Table 10-1**) are minimized when connected nodes or elements are numbered as close as possible to each other.

Prior to optimizing a model, complete all meshing operations. In addition, all coincident nodes should be merged (through Equivalencing) and the model boundaries verified. If the node or element definitions in the model are changed or modified after optimization, the model should be re-optimized.

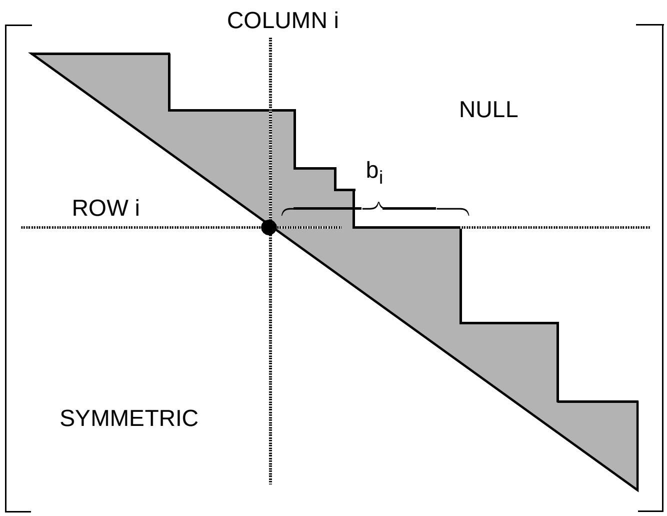


Figure 10-1 A Sparse, Symmetric Matrix

More Help:
- **Optimizing Nodes and Elements** (p. 186)
- **Selecting an Optimization Method** (p. 187)

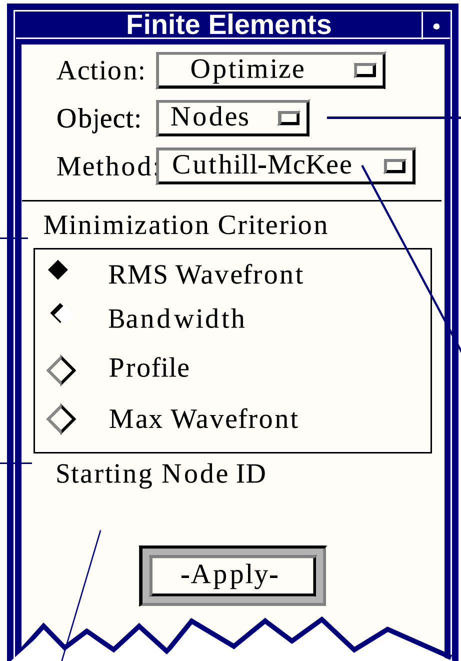
Table 10-1 The Attributes of a Matrix

Row Bandwidth	b_i = bandwidth for row i. (See **Figure 10-1** for b_i.)
Matrix Bandwidth	The matrix bandwidth, B, is given by: $B = \max b_i$.
Matrix Profile	The matrix profile, P, is given by: $P = \sum_{= 1}^{N} b_i$
Active Column	A column j is an active column in row i if there is an entry in that column in any row with index k <=1.
Row Wavefront	w_i, the row wavefront for row i, is the number of active columns in row i.
Matrix Wavefront	The matrix wavefront, W, is given by: $W = \max w_i$
RMS Wavefront	The root mean square wavefront, W_{RMS}, is given by: $W_{RMS} = (1/N)^* \sqrt{\sum_{i=1}^{N} w_i^2}$

☞More Help:

- **Optimizing Nodes and Elements** (p. 186)
- **Selecting an Optimization Method** (p. 187)

10.2 Optimizing Nodes and Elements



Either node or element IDs can be optimized.

The decision whether to choose node or element optimization is based on whether the analysis code's solution process depends on node or element order. See **Selecting an Optimization Method** (p. 187).

All nodes or elements in the model will be optimized.

Element optimization will be based on element connectivity, and node optimization will be based on node connectivity. Both elements and MPCs are taken into account when determining node connectivity.

Cuthill-McKee
Gibbs-Poole-Stk
Both

The mathematical method used to optimize nodes and elements. The Cuthill-McKee and the Gibbs-Poole-Stockmeyer methods are closely related, but may yield slightly different results for a given structure. If Both is selected, both the Cuthill-McKee and the Gibbs-Poole-Stockmeyer algorithms are invoked, and the results from the method which gives the minimum criterion value are used.

Specifies the starting ID used when renumbering the nodes or elements.

The criteria which may be minimized by the optimization algorithm are:

RMS Wavefront:The root mean square of the matrix row wavefronts

Bandwidth:The maximum matrix row bandwidth

Profile:The profile of the matrix

Max Wavefront:The maximum matrix wavefront

After Apply has been selected, MSC.Patran determines the optimal node or element ordering according to the criterion selected. All nodes or elements are then renumbered to reflect the results of the optimization. A form is displayed which lists statistics about the optimization, such as the bandwidth, profile, maximum wavefront, average wavefront, and RMS wavefront computed before and after optimization. If the "Both" method is selected the form lists after statistics for both methods.

Important: During node optimization unreferenced nodes (nodes which are not referenced by an element or an MPC) are retained and renumbered after the referenced nodes. If these nodes are not desired, they can be deleted using the Delete Nodes option.

More Help:

• **Introduction to Optimization** (p. 184)
• **Selecting an Optimization Method** (p. 187)

10.3 Selecting an Optimization Method

This section suggests the optimization type, method, and criterion to be selected for commonly used analysis codes. For analysis codes not listed below, please refer to the code vendor for a recommendation.

Note that the choice of method and criterion may depend on the structure of your model and type of analysis (static vs. dynamic). As a result, the recommendations given below are suggested only as guidelines.

Most of the commonly used analysis codes have their own built-in optimizers which internally renumber the nodes or elements. These codes are marked with an asterisk(*) in the following table. The external IDs do not change. There are a couple of advantages to using the code specific optimizers.

- They are tuned to the specific analysis code.
- They give control of the entity IDs back to the user.

However, there are cases where the MSC.Patran optimizer does a better job than the code specific optimizer.

Analysis Code	Object	Method	Minimization Criterion
ABAQUS*	Elements	BOTH	RMS WAVEFRONT
MSC.Nastran*	Nodes	BOTH	RMS WAVEFRONT
MSC.Marc*	Nodes	BOTH	RMS WAVEFRONT
FEA*	Nodes	BOTH	PROFILE

* Analysis code with built-in optimizers which internally renumber the nodes or elements.

More Help:
- Introduction to Optimization (p. 184)
- Optimizing Nodes and Elements (p. 186)

CHAPTER 11

The Verify Action

- Introduction to Verification
- Verify Forms
- Theory

11.1 Introduction to Verification

Model verification consists of a number of different tests which can be performed to check the validity of a finite element model. These tests include checks of element distortion, element duplication, model boundaries, nodal connectivity, and node/element ID numbering.

In the case of distortion checking, MSC.Patran provides a series of automated tests to measure the “distortion” of elements from an “ideal” shape through measurable geometric properties of the element. The results of these tests are compared to user specified criteria and a determination is made whether the element is acceptable or not. The pass/fail criteria is analysis code dependant and is updated automatically when the Analysis Preference is changed.

Verification tests are always performed on the current group of the active viewport except in the case of duplicate elements in which case the entire model is checked.

To get an overview when checking a specific element type, there is a test choice of *All*. When this is selected MSC.Patran will display a spreadsheet showing a summary of the total number of elements that exceed a threshold value for each of the distortion checks, and the actual test value and element ID number for the most extreme element.

Model Verification provides visual feedback of the selected test. Element distortion checks allow the selection of a threshold value using a slidebar. During the check, any element, which exceeds the threshold value, is highlighted and its value is listed in the Command Line. Upon completion of the check MSC.Patran will color code the elements based on the computed test value. Elements with a value higher than the threshold are colored with the highest spectrum color, all other values are assigned uniformly through the other spectrum levels. The current group will be rendered using the Element Fill style. Verification forms for Quad elements include an icon that allows a selection to split failed elements or simply highlight them.

Other checks, such as element duplication and connectivity, give options only to highlight any offending elements, or automatically correct the model.

Model boundaries may either be displayed as edge lines, showing unshared edges in the model, or as shaded faces, showing unshared surfaces.

All verification tests that involve color-coding, shading, or some other method of re-rendering the model have a Reset Graphics button on the form. Selecting this button will undo any rendering procedures performed by the most recent verification activity. The render style and spectrum display will be returned to the pre-test settings they had before the Apply button was selected. If you will be performing more than one type of verification test, it is recommended to choose Reset Graphics after each test is completed. Remember Reset Graphics resets to the settings prior to the current verification activity, not to those at the start of all verification.

All element specific verification forms have a *Normalize* button. By default, the normalize option will not be selected, and the slidebar will represent an actual value for the verification test threshold. If the normalize option may be selected, the slidebar will now represent a range of values from zero to one. The value of zero will represent the most reliable shape for this element type.

All element specific verification forms also have a Reset button. Selecting this button returns the slidebar and all toggles to the settings they had when the form was opened.

The information obtained from verification procedures can assist the engineer in deciding if the finite element model is satisfactory, or should be adjusted through remeshing or element modification.

11.2 Verify Forms

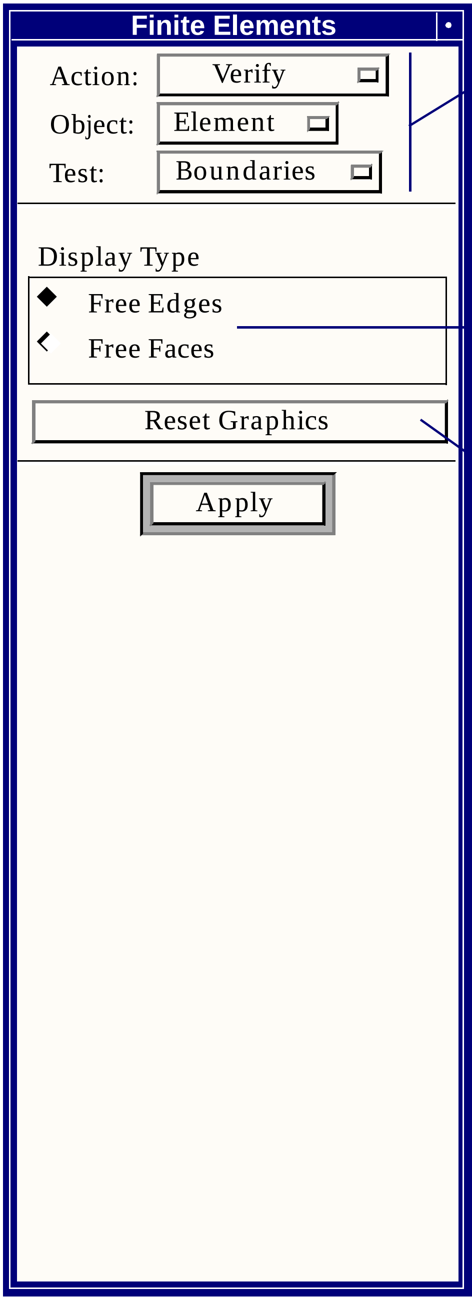
When *Verify* is the selected Action the following options are available.

Object	Test	Description
Element	**Boundaries**	Plots the free edges or faces of finite elements.
	Duplicates	Checks elements for identical corner (or end) nodes.
	Normals	Compare adjacent shell normals.
	Connectivity	Check solid elements for proper connectivity using a volume calculation.
	Geometry Fit	Checks fit error distances between elements and their parent geometry.
	Jacobian Ratio	Reports the maximum variation of the determinant of the Jacobian over each element.
	Jacobian Zero	Reports the minimum determinant of the Jacobian for each element.
	IDs	Assigns color to the Finite Elements based on the Element ID number.
Tria	**All**	Tests tria elements for each of the tria verification tests. Reports the worst case for each test and the element at which it occurs.
	Aspect	Measures length to width ratio of tria elements.
	Skew	Tests tria elements for angular deviation using an edge bisector method.
Quad	**All**	Tests quad elements for each of the quad verification tests. Reports the worst case for each test and the element at which it occurs.
	Aspect	Measures length to width ratio of quad elements.
	Warp	Tests quad elements for deviation out of plane.
	Skew	Tests quad elements for angular deviation from a rectangular shape using an edge bisector method.
	Taper	Tests quad elements for geometric deviation from a rectangular shape.

Object	Test	Description
Tet	All	Tests tet elements for each of the tet verification tests. Reports the worst case for each test and the element at which it occurs.
	Aspect	Compares ratio of height to square root of opposing face area of tet elements.
	Edge Angle	Calculates the maximum deviation angle between adjacent faces of tet elements.
	Face Skew	Tests each face of tet elements for angular deviation using an edge bisector method.
	Collapse	Tests tet elements for near zero volume.
Wedge	All	Tests wedge elements for each of the wedge verification tests. Reports the worst case for each test and the element at which it occurs.
	Aspect	Compares the maximum ratio of the height of the triangular sides to the distance between them for each wedge element.
	Edge Angle	Calculates the angular deviation between adjacent faces of wedge elements.
	Face Skew	Tests each face of wedge elements for angular deviation using an edge bisector method.
	Face Warp	Tests each quad face of wedge elements for deviation out of plane.
	Twist	Computes a twist angle between the two triangular faces of wedge elements.
	Face Taper	Tests each quad face of wedge elements for geometric deviation from a rectangular shape.
Hex	All	Tests hex elements for each of the hex verification tests. Reports the worst case for each test and the element at which it occurs.
	Aspect	Calculates the ratio of the maximum to minimum distance between opposing faces for each Hex element.
	Edge Angle	Calculates the angular deviation between adjacent faces of hex elements.
	Face Skew	Calculates the skew angle for each face of a hex element and reports the maximum.
	Face Warp	Calculates the deviation out of plane for each element face.
	Twist	Computes twist between the opposing faces of hex elements.
	Face Taper	Tests each Hex element for geometric deviation from a rectangular shape.

Object	Test	Description
Node	**IDs**	Computes contour lines based on the ID numbers of the Nodes.
Midnode	**Normal Offset**	Calculates the ratio between the perpendicular offset of the midside node and the element edge length.
	Tangent Offset	Measures the offset from the center of the element edge to the midside node. Calculates the ratio of this offset to the element edge length.
Superelement		Displays superelement’s boundaries with or without the boundary nodes.

Verify - Element (Boundaries)



Plots the boundary as free edges, or optionally free faces, of all Posted elements in all viewports. A boundary is defined as any edge or face of a finite element which is not shared by at least one other element. Therefore, this test will display, in addition to interior and exterior edges/faces of the group, any interior “cracks”. Cracks will appear along geometric boundaries prior to equivalencing.

Selecting **Free Edges** displays any unshared edges that define the boundary of your current group in yellow.

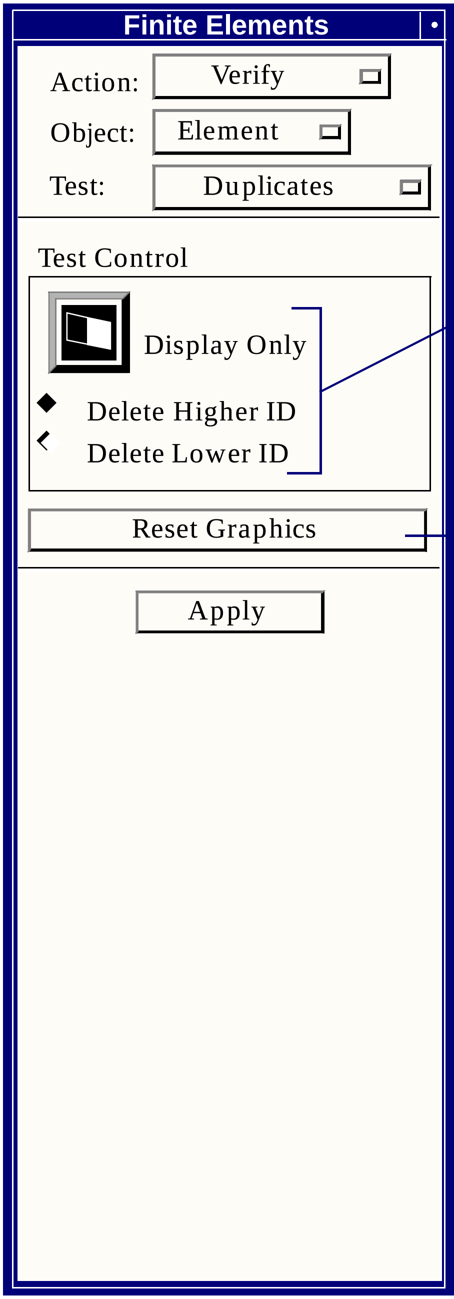
Selecting **Free Faces** displays any unshared faces in your current group in a yellow Flat Shaded render style.

Returns your graphic display to the way it was when the form was opened. This will either change from Flat Shaded (for free faces) or Boundary Line (for free edges) back to the original Render Style. If you were originally using entity type display mode, this will again become the default. Exiting this form will reset graphics.

Important: If you are in entity type display mode when you start boundary verification, MSC.Patran will temporarily enter group display mode to display the group boundaries.

Verify - Element (Duplicates)

Elements in the entire model are checked for identical corner (or end) nodes.

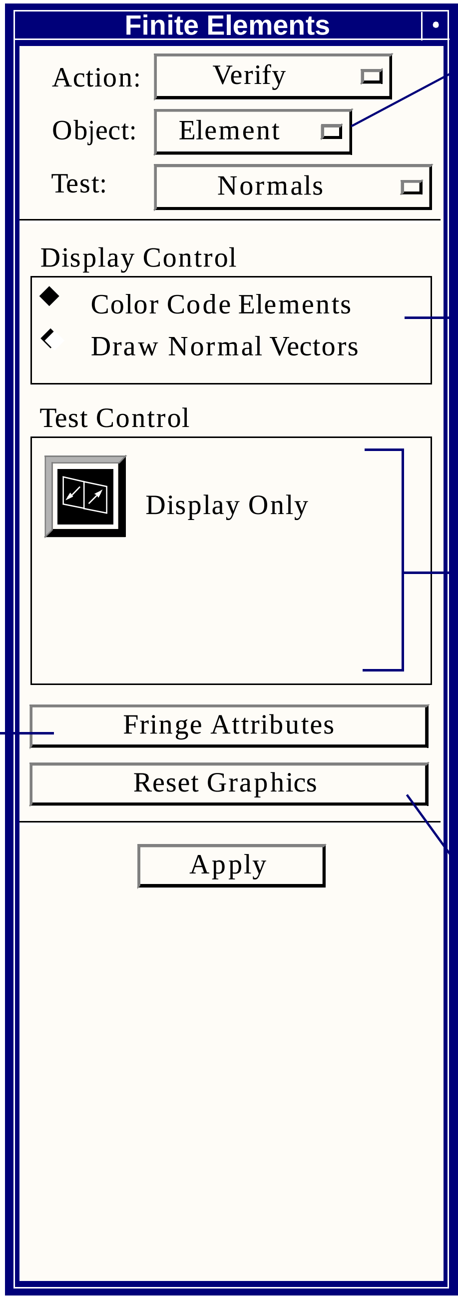


MSC.Patran gives the option to highlight any duplicate elements found, or, if you select the icon, you may choose to have MSC.Patran automatically eliminate any duplicates found. When delete duplicates is selected you may choose which element ID of the two to remove from the database.

Returns your graphic display back to the way it was when the form was opened.This will unhighlight duplicate elements. Exiting this form will also reset graphics.

Verify - Element (Normals)

Figure 11-1



Normal directions are processed for each shell element in the current group in the active viewport.

There are two options for displaying the element normal information:

1. The elements may be color-coded. Elements will initially be colored white, any element whose normal is reversed will be highlighted in red. **Note:** Be sure to equivalence your model before color coding. This is necessary because the color-coding algorithm only compares elements with shared edges.
2. Or MSC.Patran may draw the normal vectors and plot arrows pointing from the element centroids in the element normal direction.

There are two available options for controlling what MSC.Patran does with the element normal information:

1. The default choice is Display Only which tells MSC.Patran the only action you want it to take is to graphically display the normal information.
2. Or you may select the Display Only icon to change the test control to Reverse Elements. This allows you to specify (or select) a Guiding Element to which all other element normals in your current group will be matched.

Returns your graphic display to the way it was when the form was opened. This will usually change from Element Fill back to the original Render Style, or will remove all those little vectors from the viewport. Exiting this form will also reset graphics.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

Verify - Element (Connectivity)

All solid elements in the current group in the active viewport are checked for proper connectivity using a volume calculation.

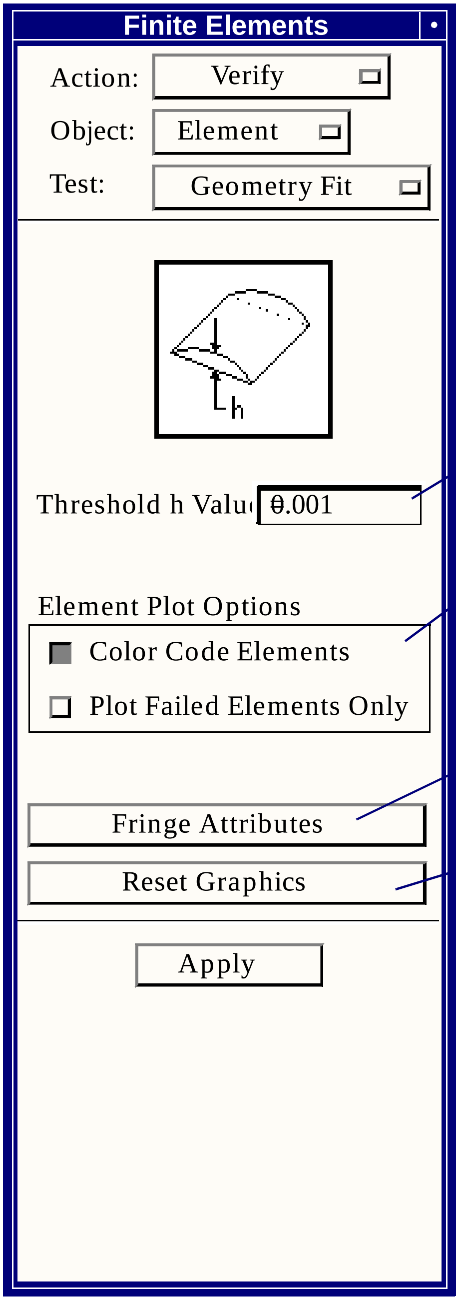
Elements found to have negative volume will be color coded for identification purposes when the highlight only icon is selected. Otherwise, the offending elements will be automatically reversed when the reverse icon is selected.

More Help:

• **MSC.Patran's Element Library** (p. 378)

Verify - Element (Geometry Fit)

All elements in the current group in the active viewport are checked for maximum distance between the element and the parent geometry.



Enter the maximum acceptable distance between an element and its parent geometry.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group that were tested and failed.

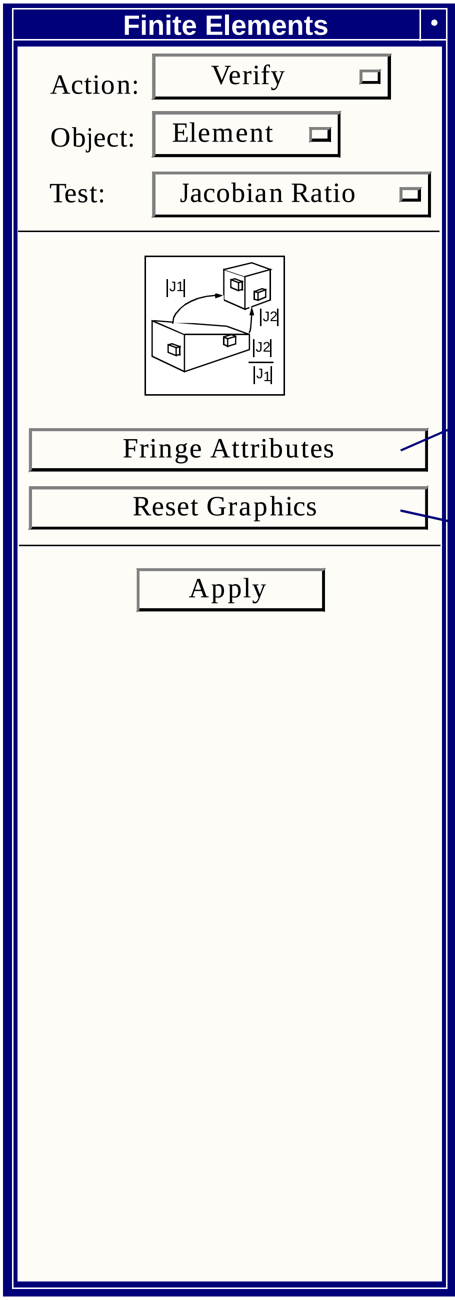
See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

Returns your graphic display back to the way it was when the form was opened. This will usually change from Element Fill back to the original Render Style and remove the spectrum display from your viewport.

Note: Linear elements such as Bar/2, Quad/4, and Hex/8 are evaluated at one point per bar or element face. Quadratic elements such as Bar/3, Quad/8, and Hex/20 are evaluated at two points per bar or four points per element face. Cubic elements such as Bar/4, Quad/12, and Hex/32 are evaluated at three points per bar or nine points per element face.

Verify - Element (Jacobian Ratio)

The ratio of the maximum determinant of the Jacobian to the minimum determinant of the Jacobian is calculated for each element in the current group in the active viewport. This element shape test can be used to identify elements with interior corner angles far from 90 degrees or high order elements with misplaced midside nodes. A ratio close or equal to 1.0 is desired.



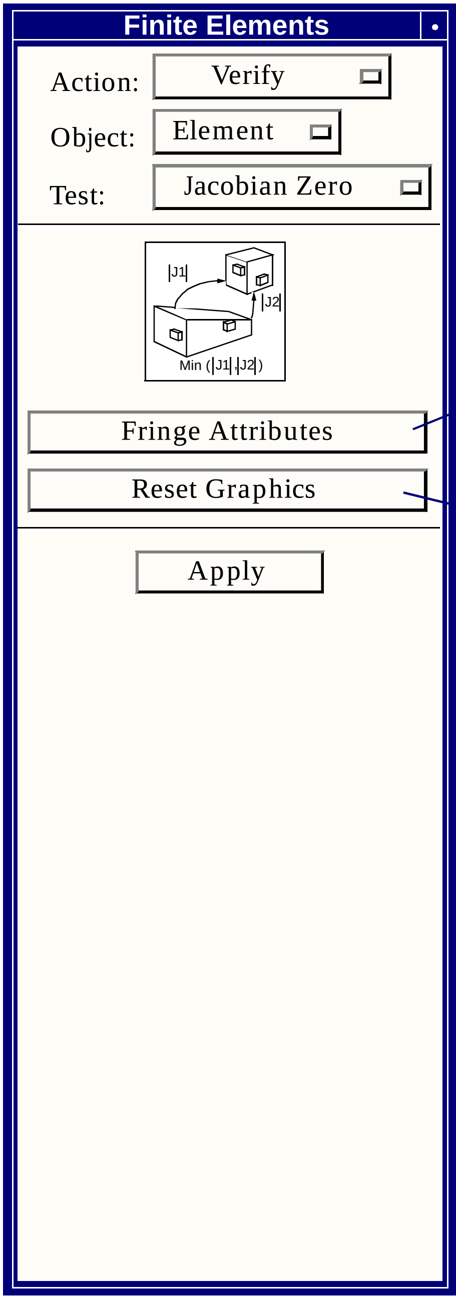
See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

Returns your graphic display to the way it was when you entered the form. This will usually change from Element Fill back to the original Render Style, and remove the spectrum display from your viewport.

> **Note:** The minimum and maximum ratios and the associated elements are echoed in the command line. Elements in the current group are color-coded according to the value of the Jacobian ratio and will be plotted in the Element Fill render style.

Verify - Element (Jacobian Zero)

The determinant of the Jacobian (J) is calculated at all integration points for each element in the current group in the active viewport. The minimum value for each element is determined. This element shape test can be used to identify incorrectly shaped elements. A well-formed element will have J positive at each Gauss point and not greatly different from the value of J at other Gauss points. J approaches zero as an element vertex angle approaches 180 degrees.



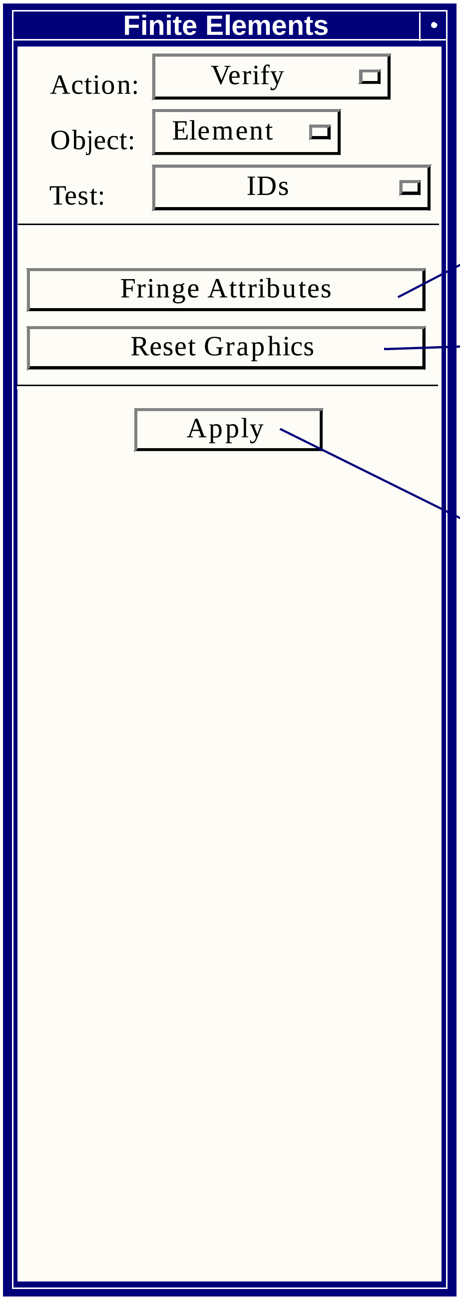
See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

Returns your graphic display to the way it was when the form was opened. This will usually change from Element Fill back to the original Render Style, and remove the spectrum display from your viewport.

Note: The minimum and maximum value and the associated elements are echoed in the command line. Elements in the current group are color-coded according to the value of the determinant of the Jacobian and will be plotted in the Element Fill render style.

Verify - Element (IDs)

Each element in the current group in the active viewport is assigned a color based on its ID number.



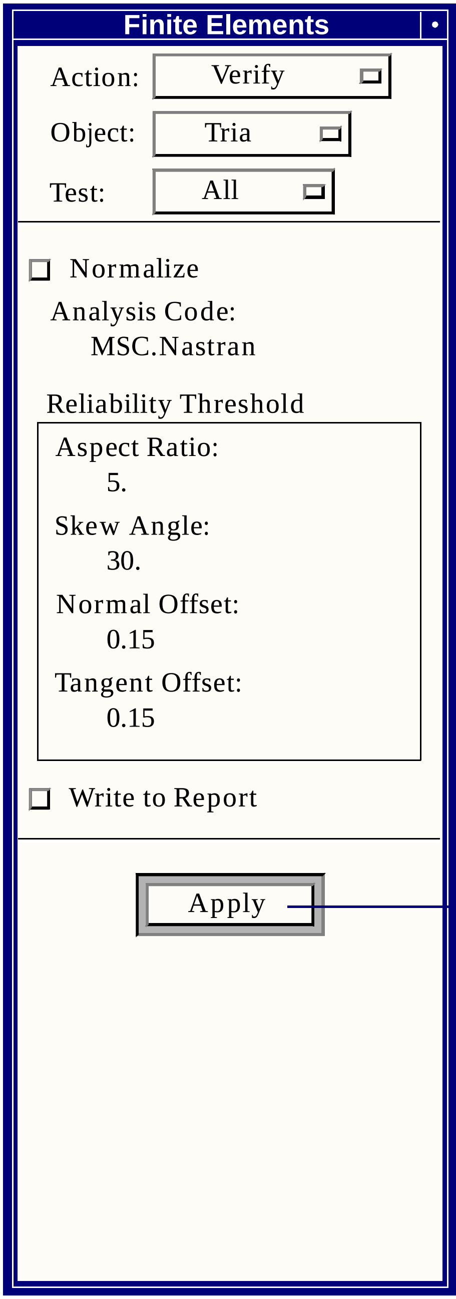
See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

Returns your graphic display to the way it was when the form was opened. This will usually turn off the Spectrum Display, and change from Element Fill back to your original Render Style.

When the Apply button is selected, the color assignments are generated. The model is plotted with the elements on the visible surfaces filled in with their corresponding color.

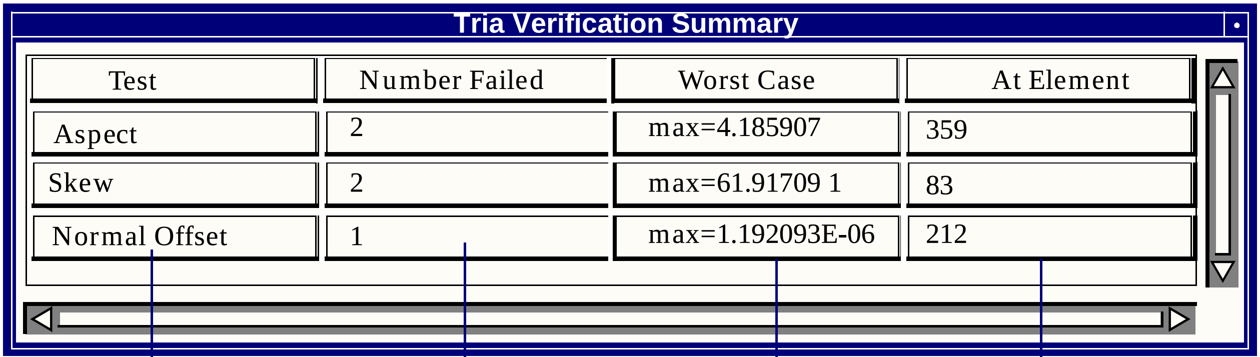
Verify - Tria (All)

Each tria element in the current group is tested for each of the tria verification tests.



When the Apply button is selected, a spreadsheet will be displayed showing the worst case value for each test and the element at which it occurs.

Verify - Tria (All) Spreadsheet



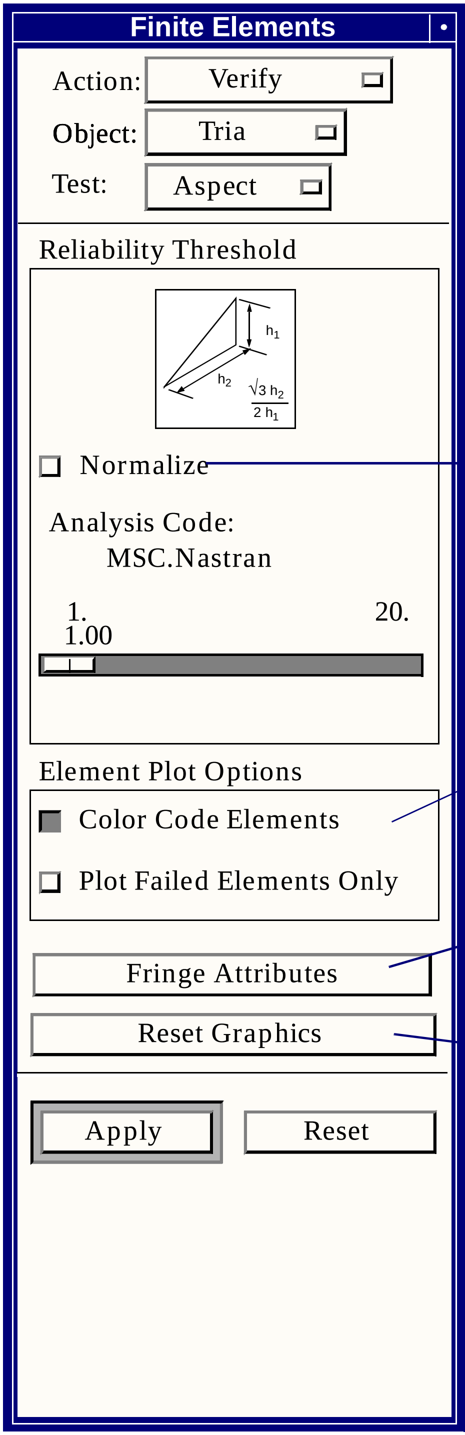
This column lists the Verification test. All tests listed on the application form will be performed. Use the scroll bar to view additional tests.

This column lists the number of elements in the current group that exceeded the reliability threshold shown on the application form.

The last two columns list the maximum (or minimum, if applicable) value of the test and the element at which this worst case occurs.

Verify - Tria (Aspect)

All of the tria elements in the current group of the active viewport are tested for length to width ratio.



When the Normalize button is selected, the computed aspect ratio is inverted and subtracted from 1. When the Normalize button is turned OFF, the slidebar represents the computed Aspect Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

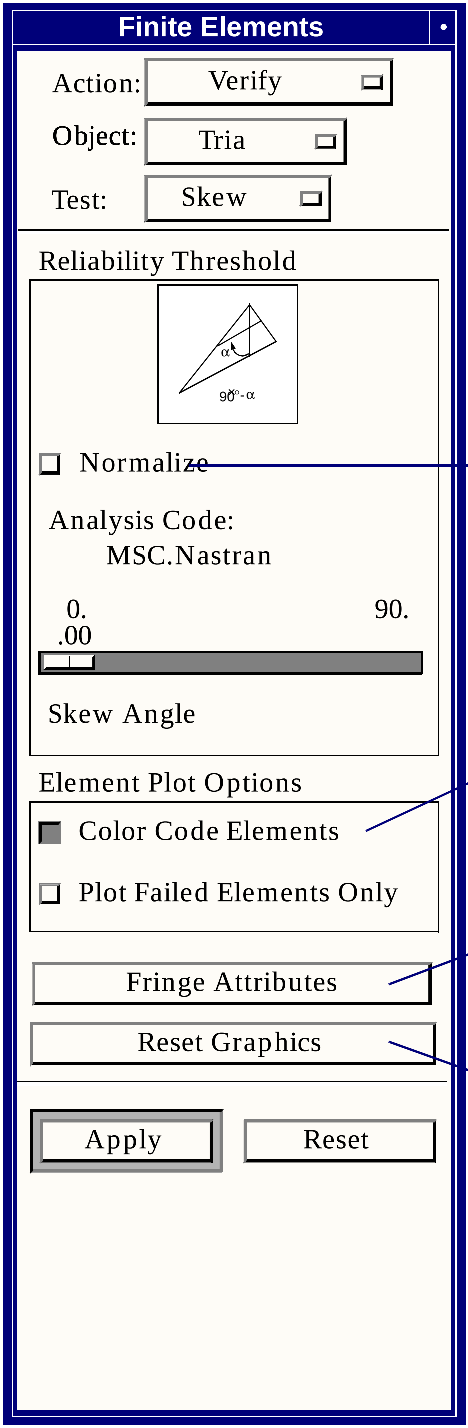
Returns your graphic display to the way it was when the form was opened. This will usually turn off the Spectrum Display, and change from Element Fill back to your original Render Style.

More Help:

• **Aspect Ratio** (p. 241)

Verify - Tria (Skew)

Each tria element in the current group of the active viewport is tested for skew. The skew angle is obtained as described in **Theory** (p. 238).



When the Normalize button is selected, the computed skew angle will be divided by 90°. An equilateral triangle will have a skew angle (and skew factor) of 0. When the Normalize button is turned off, the slidebar represents the computed Skew Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

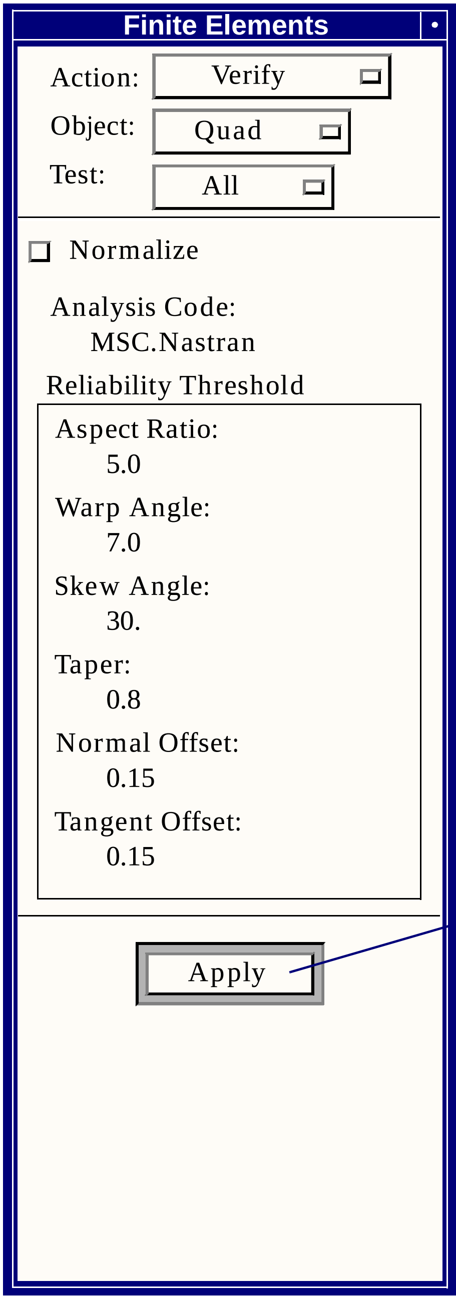
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display, and change from Element Fill back to your original Render Style.

More Help:

- How Skew Angle is computed (p. 238)

Verify - Quad (All)

Each quad element in the current group of the active viewport is tested for each of the quad verification tests.

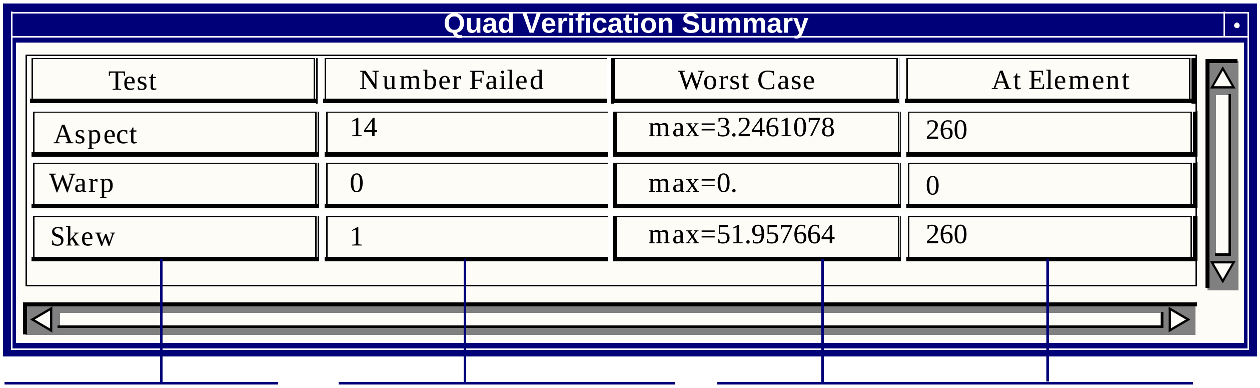


When the Apply button is selected a spreadsheet will be displayed showing the worst case value for each test and the element at which it occurs.

More Help:

- Spreadsheet Information (p. 207)
- Test Definitions (p. 238)

Verify - Quad (All) Spreadsheet



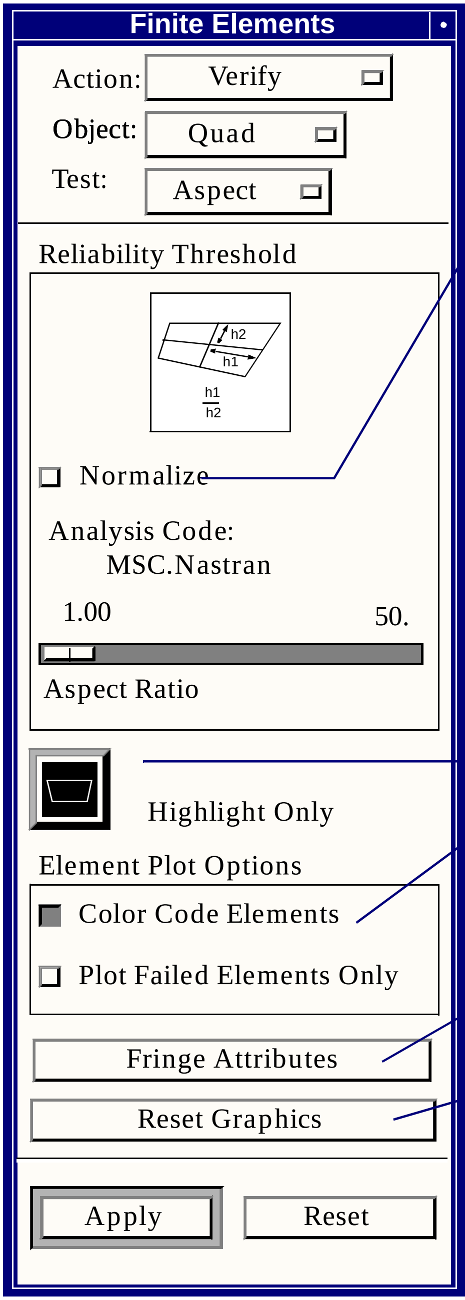
This column lists the Verification test. All tests listed on the application form will be performed. Use the scroll bar to view additional tests.

This column lists the number of elements in the current group that exceeded the reliability threshold shown on the application form.

The last two columns list the maximum (or minimum, if applicable) value of the test and the element at which this worst case occurs.

Verify - Quad (Aspect)

All of the quads in the current group, in the active viewport, are tested for length to width ratio. During the check, if an element exceeds the threshold value set by the slidebar, it is highlighted and MSC.Patran echoes the element's ID number, and its aspect value in the command line. At completion, each element is color-coded according to the value computed for its aspect value, and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed aspect ratio will be subtracted from 1. A square element with equal edge lengths will have a normalized value equal to zero $\left(1 - \frac{h_1}{h_2}\right)$ and therefore, will be the most reliable.

When the Normalize button is turned Off, the slidebar represents the computed Aspect Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Elements that exceed the aspect ratio specified by using the slidebar, will be color coded for identification purposes when the highlight only icon is selected. Otherwise, the offending elements will be highlighted in the viewport one-by-one, and the user will be prompted for permission to split each element into two elements.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

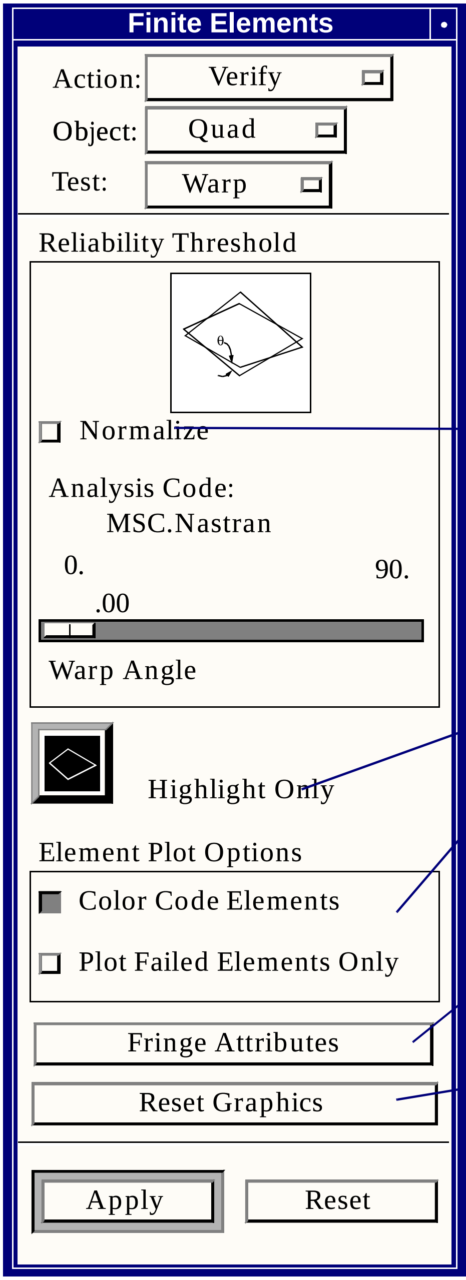
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to the original Render Style.

More Help:

• How Aspect Ratio is computed (p. 241)

Verify - Quad (Warp)

Each quad element in the current group of the active viewport is tested for warp. The warp angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its warp value in the command line. At completion, each element is color-coded according to the value computed for its warp value and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed warp angle will be divided by 15°. A "perfect" element will have no out-of-plane component and will therefore be the most reliable. When the Normalize button is turned off, the slidebar represents the computed Warp Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Elements that exceed the warp angle specified by using the slidebar, will be color coded for identification purposes when the highlight only icon is selected. Otherwise, the offending elements will be highlighted in the viewport one-by-one, and the user will be prompted for permission to split each element into two tria elements.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*

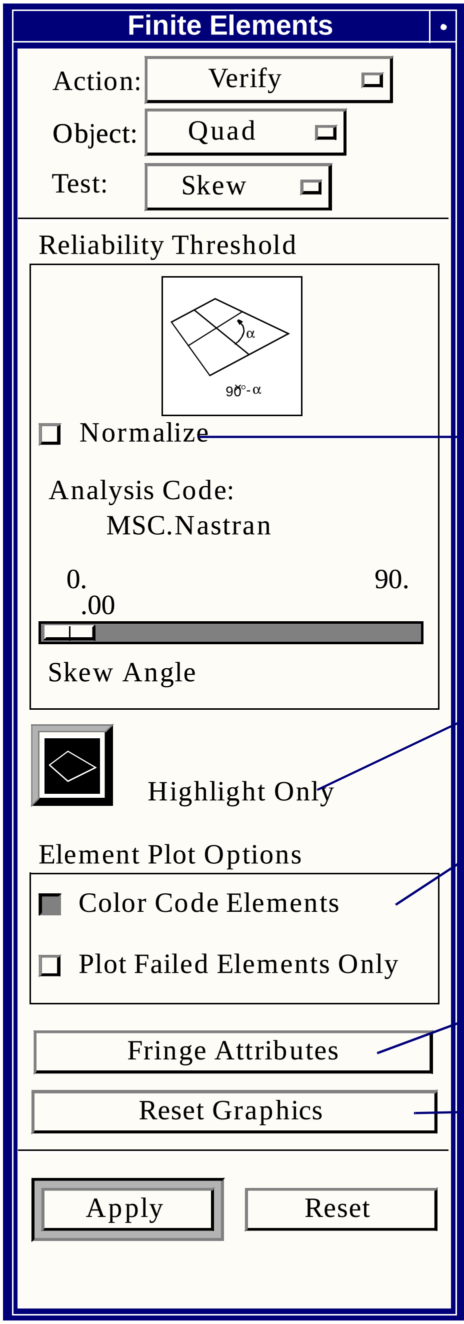
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Warp Angle is computed (p. 245)

Verify - Quad (Skew)

Each quad element in the current group in the active viewport is tested for skew. The skew angle is obtained as described in **Theory** (p. 238). Prior to testing for skew, each element is checked for convexity. If any element fails the convexity, test a warning message will be issued. Processing will continue on to the next element.



When the Normalize button is selected the computed skew angle will be divided by 90°. A "perfect" element with corner angles of 90° will have a normalized value equal to zero $\left(\frac{90° - \alpha}{90°}\right)$ and will therefore be the most reliable. When the Normalize button is turned OFF, the slidebar represents the computed Skew Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Elements that exceed the skew angle specified by using the slidebar, will be color coded for identification purposes when the highlight only icon is selected. Otherwise, the offending elements will be highlighted in the viewport one-by-one, and the user will be prompted for permission to split each element into two tria elements.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

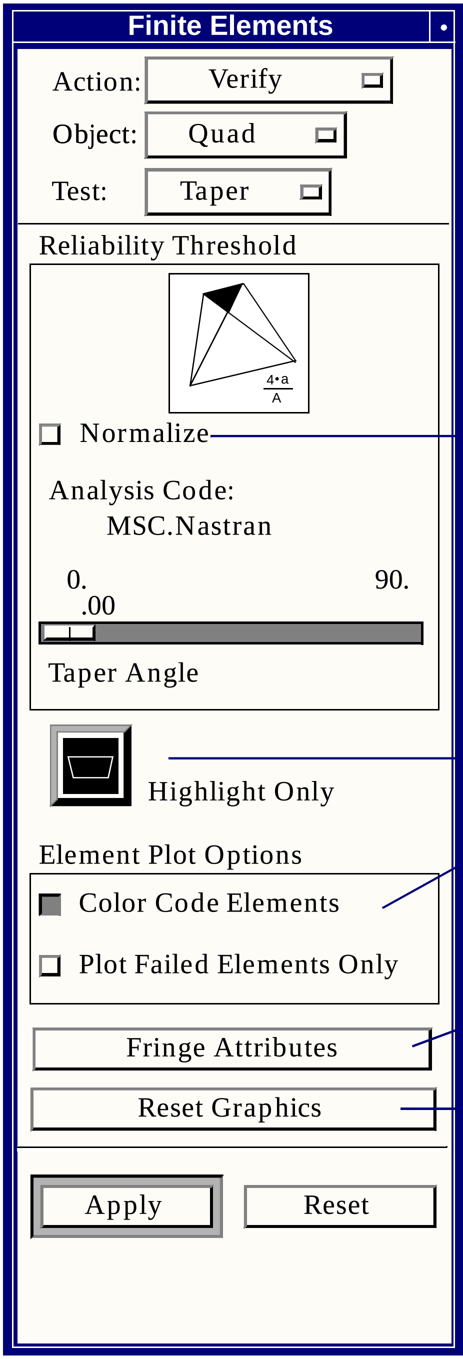
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Skew Angle is computed (p. 238)

Verify - Quad (Taper)

Each quad element in the current group in the active viewport is tested for taper. The taper ratio is obtained as described in **Theory** (p. 238).



When the Normalize button is selected, the computed Taper ratio will be subtracted from 1. A "perfect" element will have all 4 triangular subareas (a) equal; therefore, 4*a/total area will equal 1 and the normalized equation will yield a taper factor of 0. When the Normalize button is turned OFF, the slidebar represents the computed Taper Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Elements that exceed the Taper ratio specified by using the slidebar, will be color coded for identification purposes when the highlight only icon is selected. Otherwise, the offending elements will be highlighted in the viewport one-by-one, and the user will be prompted for permission to split each element into two quad elements.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

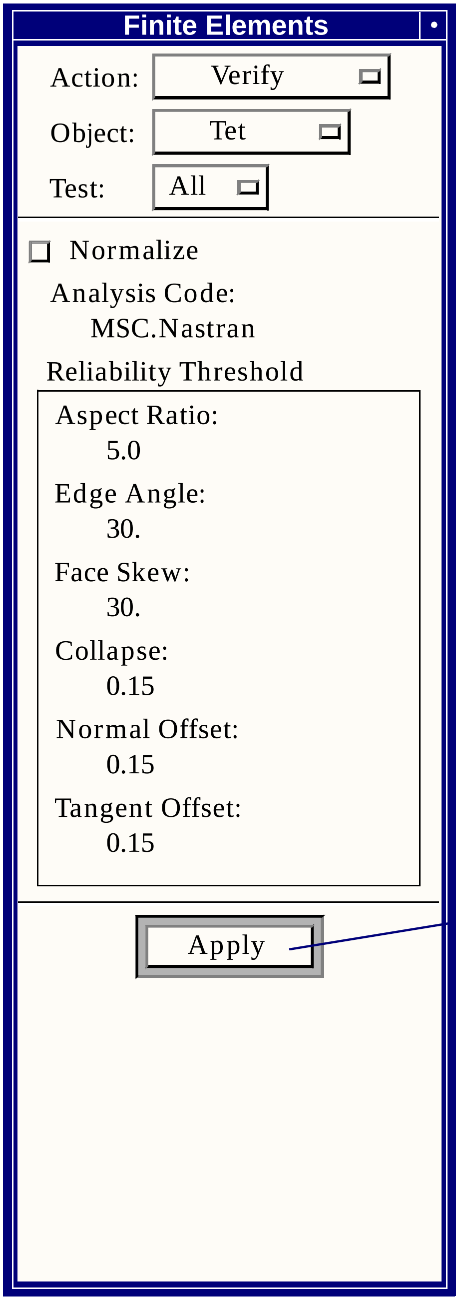
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Taper is computed (p. 246)

Verify - Tet (All)

Each Tetrahedral element in the current group is tested for each of the Tet verification tests.

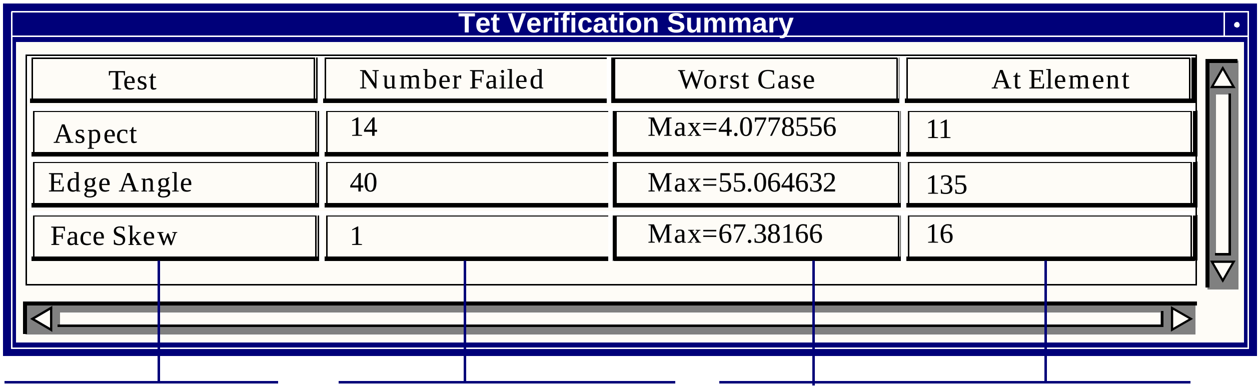


When the Apply button is selected, a spreadsheet will be displayed showing the worst case value for each test and the element at which it occurs.

More Help:

- Spreadsheet Information (p. 213)
- Test Definition (p. 238)

Verify - Tet (All) Spreadsheet



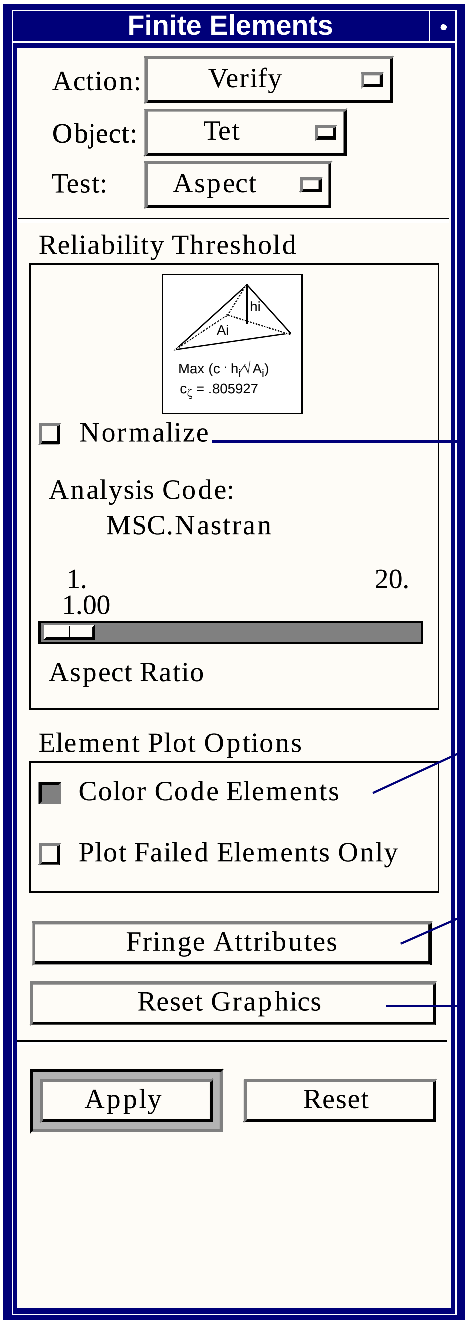
This column lists the Verification test. All tests listed on the application form will be performed. Use the scroll bar to view additional tests.

This column lists the number of elements in the current group that exceeded the reliability threshold shown on the application form.

The last two columns list the maximum (or minimum, if applicable) value of the test and the element at which this worst case occurs.

Verify - Tet (Aspect)

All of the Tets in the current group are tested for the ratio of height to square root of opposing face area. During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its aspect ratio in the command line. At completion, each element is color-coded according to the value computed for its aspect ratio and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed aspect ratio will be subtracted from 1. When the Normalize button is turned OFF, the slidebar represents the computed Aspect Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

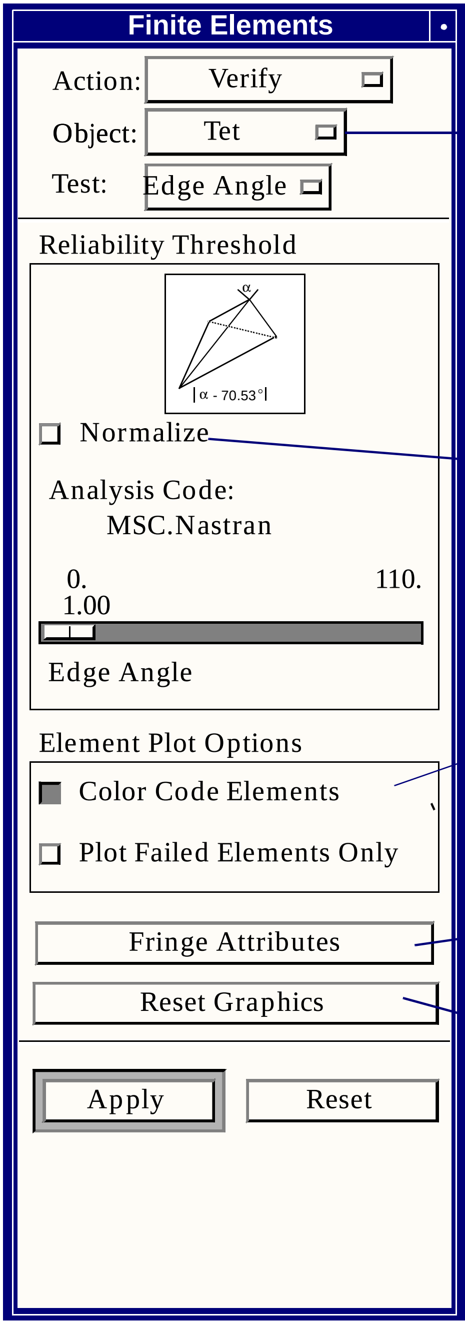
See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Aspect Ratio is computed (p. 242)

Verify - Tet (Edge Angle)



All of the tetrahedral elements in the current group of the active viewport are tested for the maximum angle between adjacent faces. An edge angle is the absolute value of the angle between the two faces meeting at an edge subtracted from 70.529°. During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its edge angle in the command line. At completion, each element is color-coded according to the edge angle and the current group is plotted in the Element Fill Render style.

When the Normalize button is selected, the computed edge angle will be divided by 110°. When the Normalize button is turned OFF, the slidebar represents the computed Edge Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

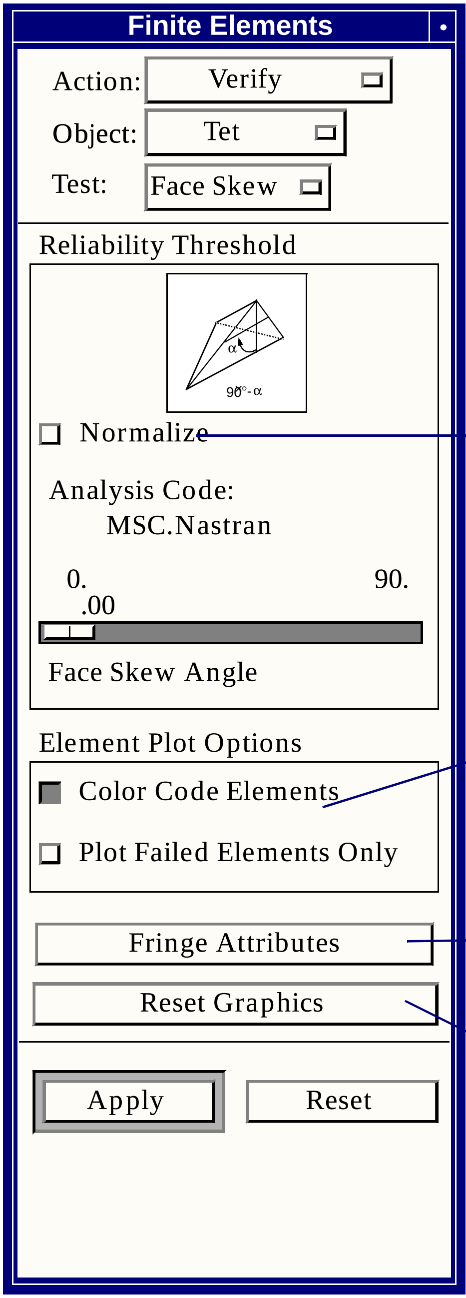
Returns your graphic display to the way it was when the form was opened. This will usually turn off the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Edge Angle is computed (p. 247)

Verify - Tet (Face Skew)

Each face of each tetrahedral element in the current group is tested for skew as if it were a Tria element. The skew angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element’s ID number and its maximum skew angle in the command line. At completion, each element is color-coded according to the value computed for its skew angle and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed skew angle will be divided by 90°. An equilateral triangle will have a skew angle of 0. When the Normalize button is turned OFF, the slidebar represents the computed skew angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

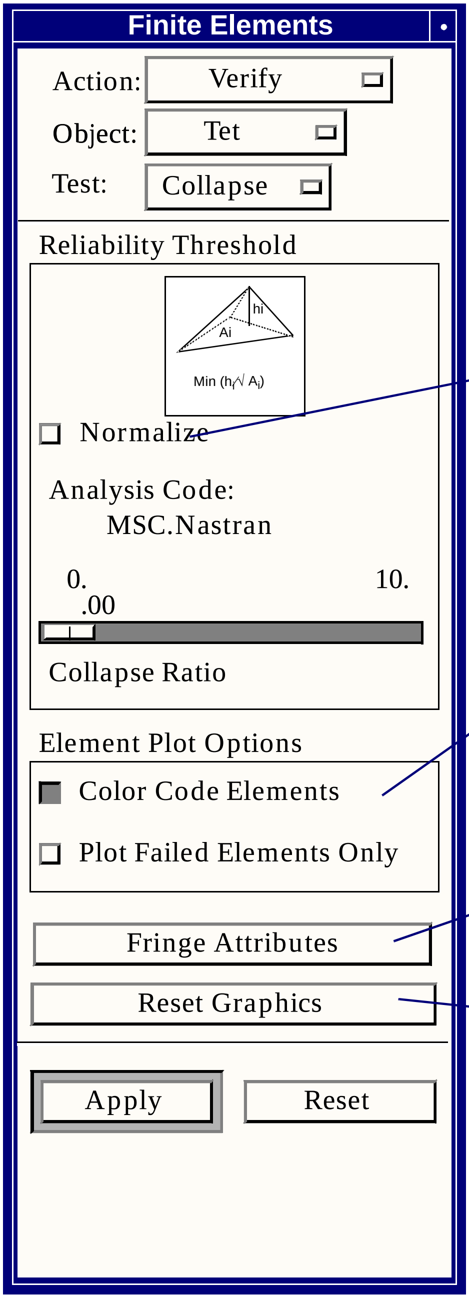
Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn off the Spectrum Display and change from Element Fill back to your original Render Style.

Verify - Tet (Collapse)

All of the tetrahedral elements in the current group are tested for volume. The collapse value is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its collapse value in the command line. At completion, each element is color-coded according to the collapse value and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed collapse ratio will be subtracted from 1. For an equilateral tet, the collapse value will equal 1, therefore,

$1 - \mathrm{Min}(h_i / (\sqrt{A_i}))$ will equal zero, and will be the most reliable element. When the Normalize button is turned Off, the slidebar represents the computed collapse value. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

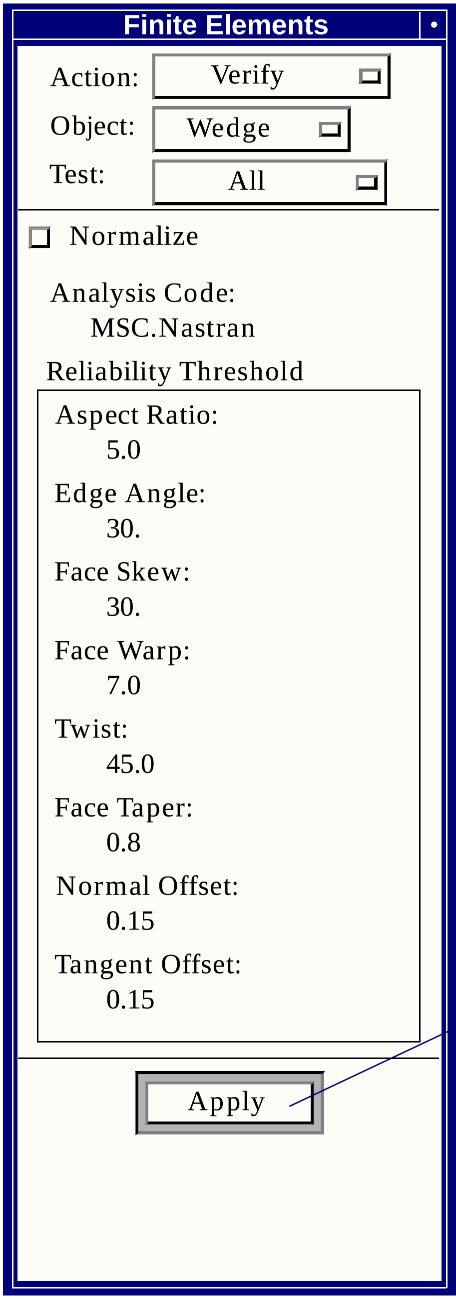
Returns your graphic display to the way it was when the form was opened. This will usually turn off the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Collapse is computed (p. 249)

Verify - Wedge (All)

Each wedge element in the current group of the active viewport is tested for each of the wedge verification tests.

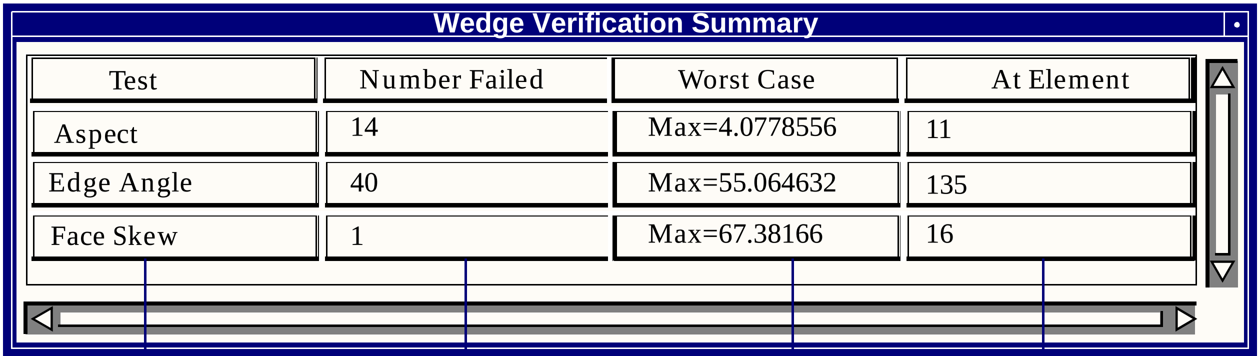


When the Apply button is selected, a spreadsheet will be displayed showing the worst case value for each test and the element at which it occurs.

More Help:

- Spreadsheet Information (p. 219)
- Test Definitions (p. 238)

Verify - Wedge (All) Spreadsheet



Wedge Verification Summary

Test	Number Failed	Worst Case	At Element
Aspect	14	Max=4.0778556	11
Edge Angle	40	Max=55.064632	135
Face Skew	1	Max=67.38166	16

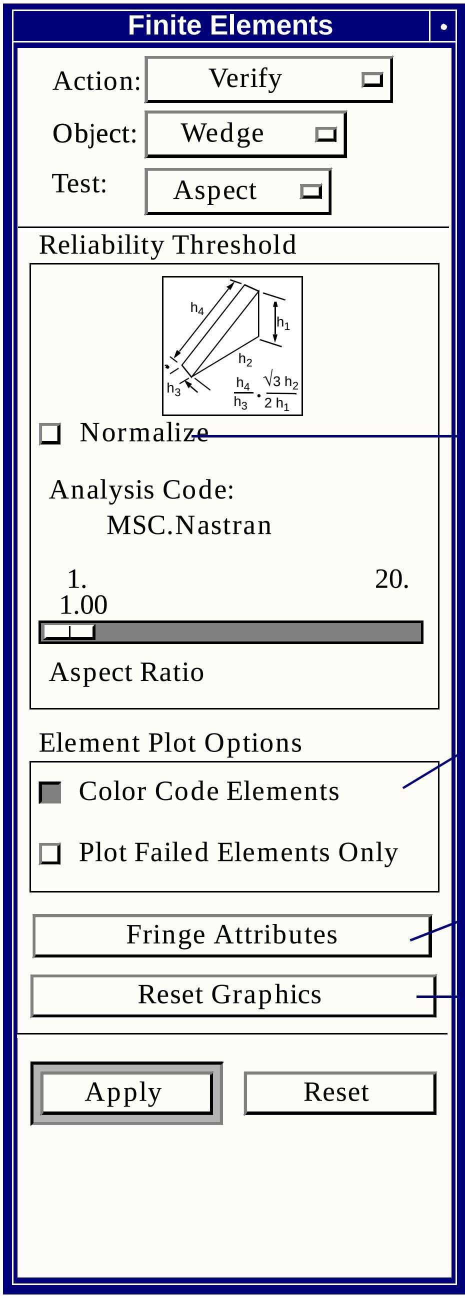
This column lists the Verification test. All tests listed on the application form will be performed. Use the scroll bar to view additional tests.

This column lists the number of elements in the current group that exceeded the reliability threshold shown on the application form.

The last two columns list the maximum (or minimum, if applicable) value of the test and the element at which this worst case occurs.

Verify - Wedge (Aspect)

All of the wedge elements in the current group are tested for length to width ratio. During the check, if an element exceeds the threshold value set by the slidebar, it is highlighted and MSC.Patran echoes the element's ID number and its aspect value in the command line. At completion, each element is color-coded according to the value computed for its aspect value and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed aspect ratio will be subtracted from 1. When the Normalize button is turned OFF, the slidebar represents the computed Aspect Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

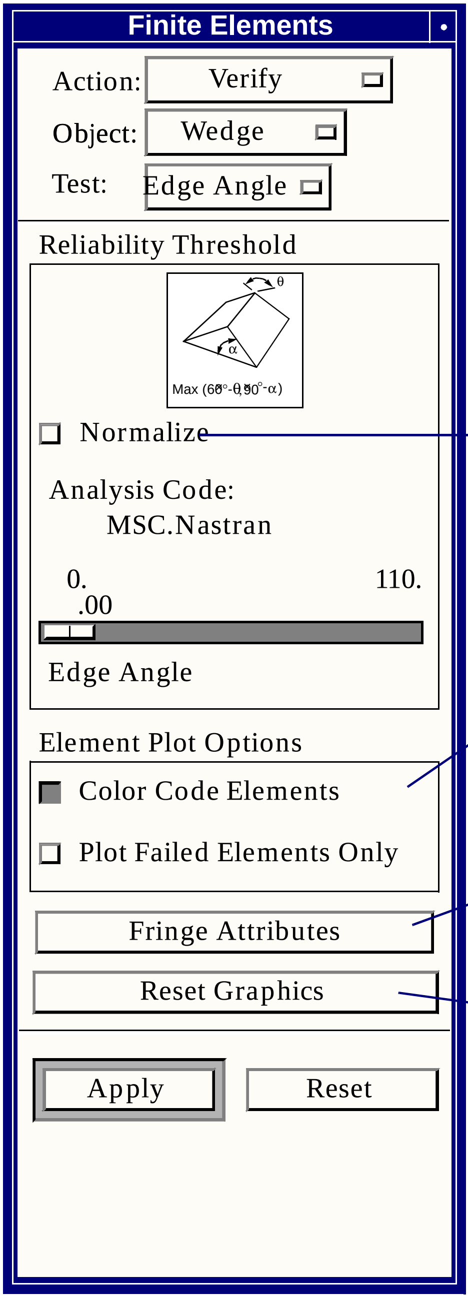
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Aspect Ratio is computed (p. 243)

Verify - Wedge (Edge Angle)

The maximum edge angle is calculated for each wedge element in the current group of the active viewport. Edge angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum edge angle in the command line. At completion, each element is color-coded according to the value computed for its edge angle and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed Edge Angle will be divided by 110°. When the Normalize button is turned OFF, the slidebar represents the computed Edge Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

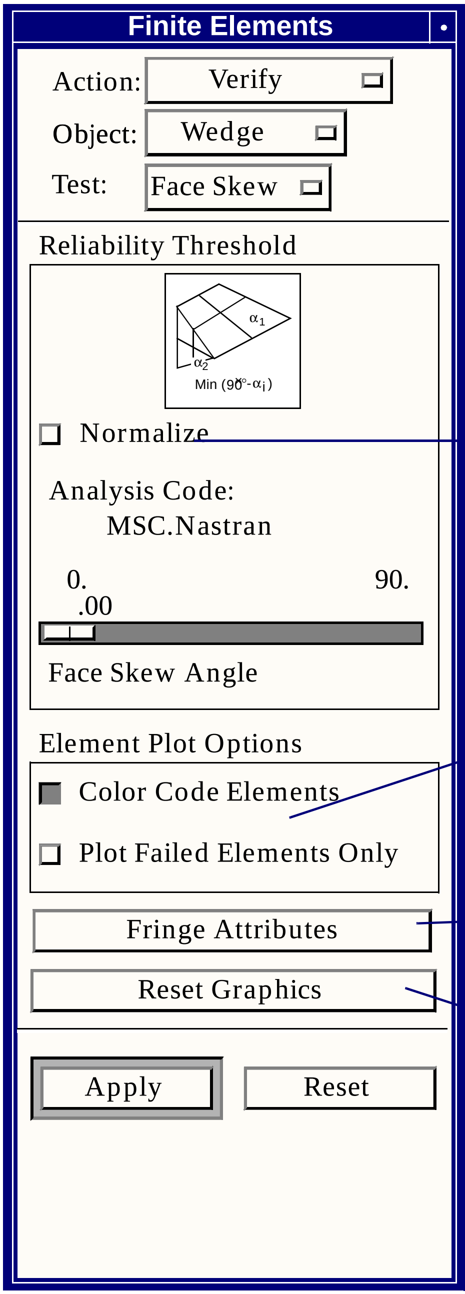
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Edge Angle is computed (p. 248)

Verify - Wedge (Face Skew)

Each face of each wedge element in the current group is tested for skew as if it were a quad or tria element. The skew angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum skew angle in the command line. At completion, each element is color-coded according to the value computed for its skew angle and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed skew angle will be divided by 15°. When the Normalize button is turned OFF, the slidebar represents the computed Skew Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

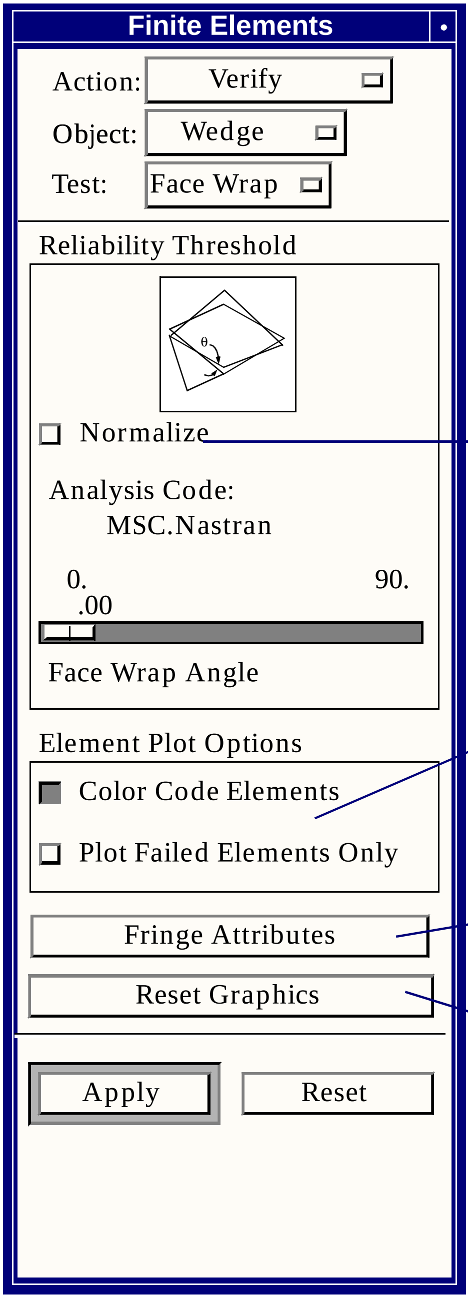
Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

Verify - Wedge (Face Warp)

Each quad face of each wedge element in the current group is tested for warp as if it were a quad. The warp angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its warp value in the command line. At completion, each element is color-coded according to the value computed for its warp value and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed warp angle will be divided by 15°. A "perfect" element will have no out-of-plane component and will therefore be the most reliable. When the Normalize button is turned OFF, the slidebar represents the computed Face Warp Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

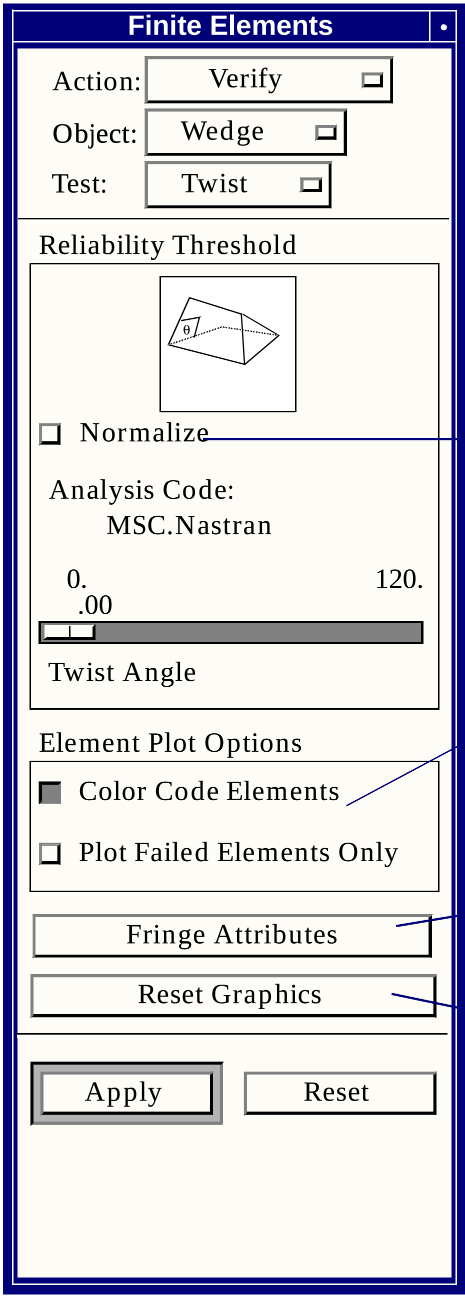
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Warp Angle is computed (p. 245)

Verify - Wedge (Twist)

Each wedge element in the current group is tested for a twist angle computed between its two triangular faces. Twist is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum twist in the command line. At completion, each element is color-coded according to the value computed for its twist and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed twist angle will be divided by 60°. When the Normalize button is turned OFF, the slidebar represents the computed Twist Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

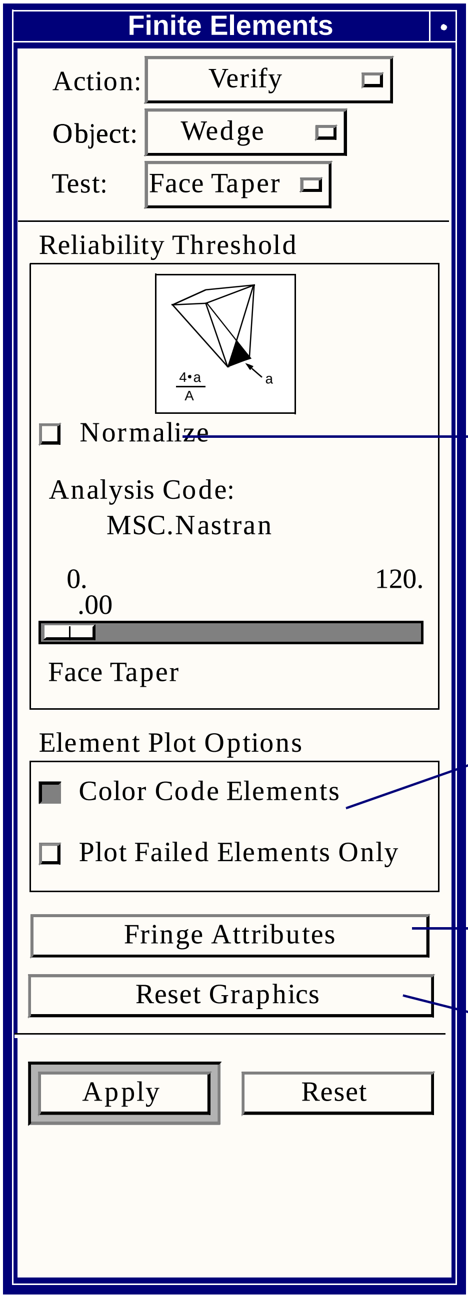
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Twist is computed (p. 250)

Verify - Wedge (Face Taper)

Each quad face of each wedge element in the current group of the active viewport is tested for taper as if it were a Quad element. Taper is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum taper in the command line. At completion, each element is color-coded according to the value computed for its taper and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed Taper ratio will be subtracted from 1. A "perfect" face will have all 4 triangular subareas (a) equal, therefore, 4*a/total area will equal 1 and the normalized equation will yield a Taper Factor of 0. When the Normalize button is turned OFF, the slidebar represents the computed Face Taper. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

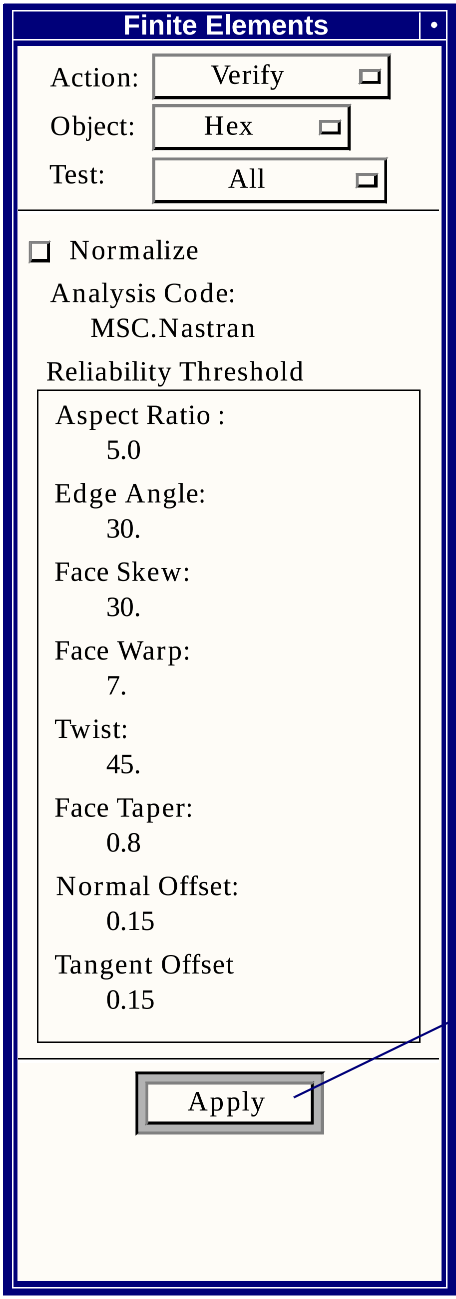
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Face Taper is computed (p. 246)

Verify - Hex (All)

Each hex element in the current group of the active viewport is tested for each of the hex verification tests.

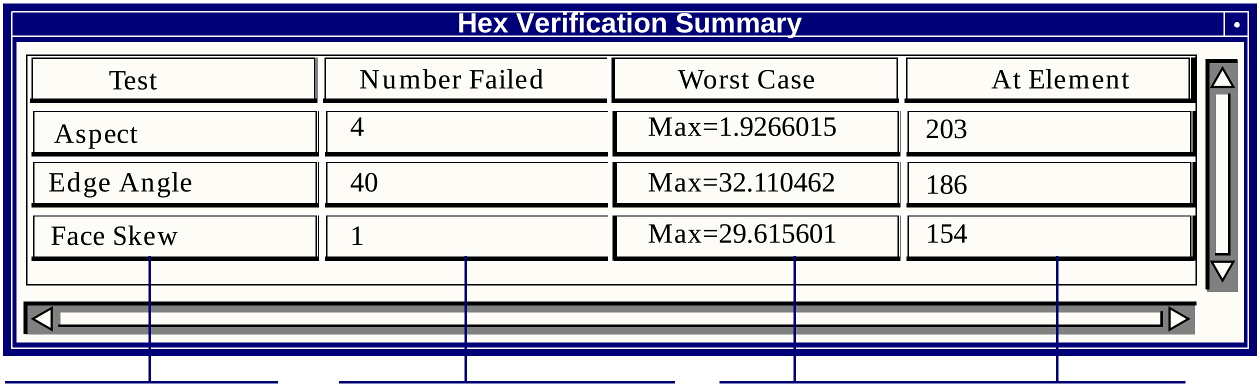


When the Apply button is selected, a spreadsheet will be displayed showing the worst case value for each test and the element at which it occurs.

More Help:

- Spreadsheet Information (p. 227)
- Test Definitions (p. 238)

Verify - Hex (All) Spreadsheet



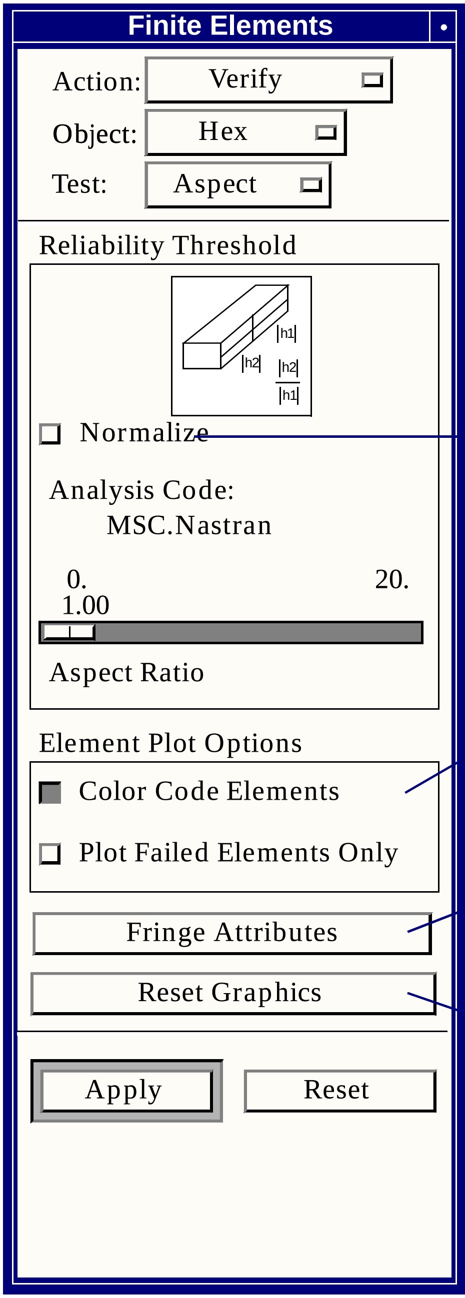
This column lists the Verification test. All tests listed on the application form will be performed. Use the scroll bar to view additional tests.

This column lists the number of elements in the current group that exceeded the reliability threshold shown on the application form.

The last two columns list the maximum (or minimum, if applicable) value of the test and the element at which this worst case occurs.

Verify - Hex (Aspect)

All of the hex elements in the current group in the active viewport are tested for length to width ratio. During the check, if an element exceeds the threshold value set by the slidebar, it is highlighted and MSC.Patran echoes the element’s ID number and its aspect value in the command line. At completion, each element is color-coded according to the value computed for its aspect value and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed aspect ratio will be subtracted from 1. When the Normalize button is turned OFF, the slidebar represents the computed Aspect Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

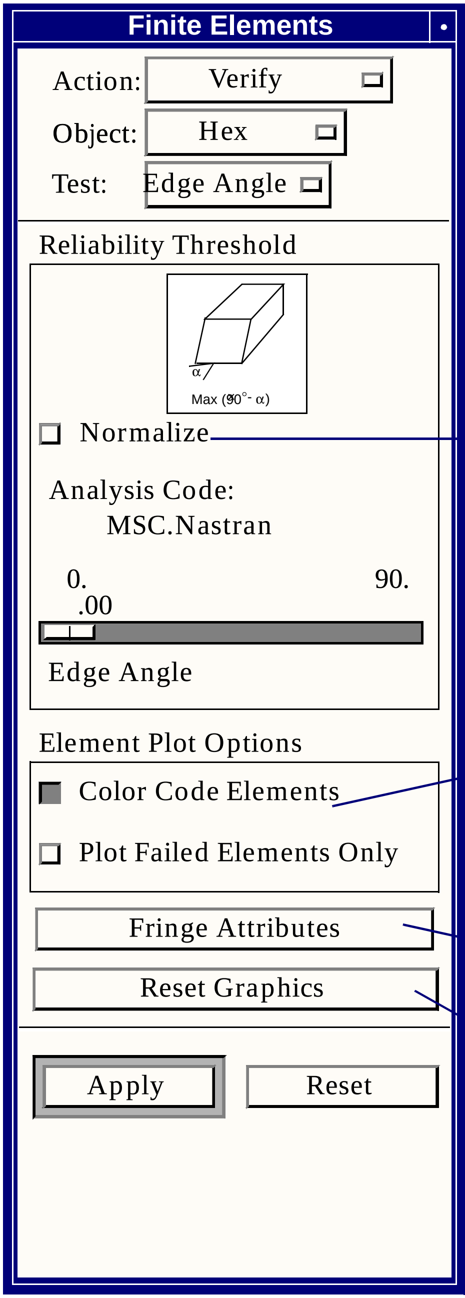
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Aspect Ratio is computed (p. 244)

Verify - Hex (Edge Angle)

The maximum edge angle is calculated for each hex element in the current group. Edge angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum edge angle in the command line. At completion, each element is color-coded according to the value computed for its edge angle and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed taper ratio will be divided by 90°. When the Normalize button is turned OFF, the slidebar represents the computed Edge Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

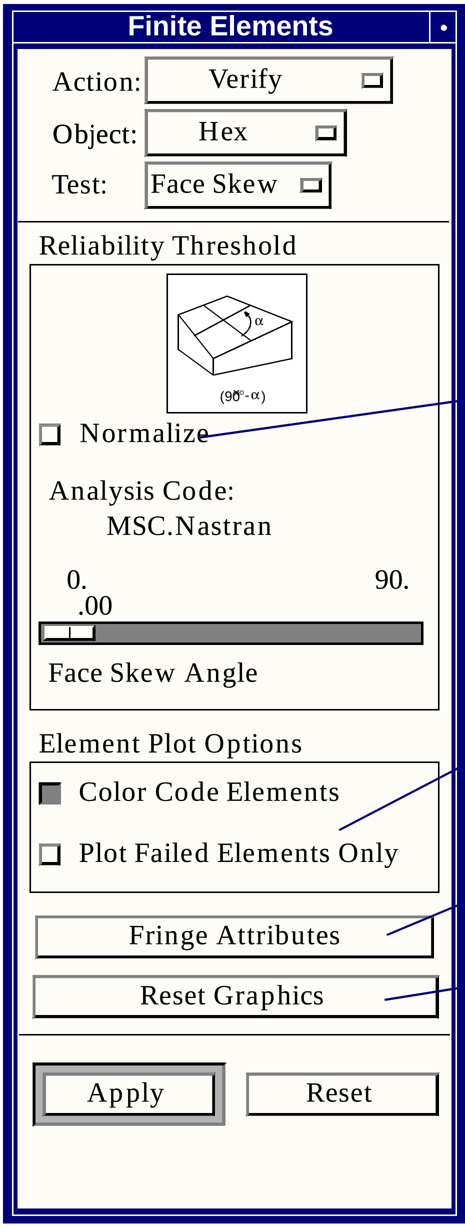
Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

Verify - Hex (Face Skew)

Each face of each hex element in the current group is tested for skew as if it were a quad element. The skew angle is obtained as described in **Theory** (p. 238). Prior to testing for skew, each element face is checked for convexity. If any face fails the convexity test, a warning message will be issued. Processing will continue on to the next element face. During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum skew angle in the command line. At completion, each element is color-coded according to the value computed for its skew angle and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed skew angle will be divided by 90°. A "perfect" element with corner angles of 90° will have a normalized value equal to zero $\left(\frac{90^\circ - \alpha}{90^\circ}\right)$ and will therefore be the most reliable. When the Normalize button is turned OFF, the slidebar represents the computed Face Skew Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

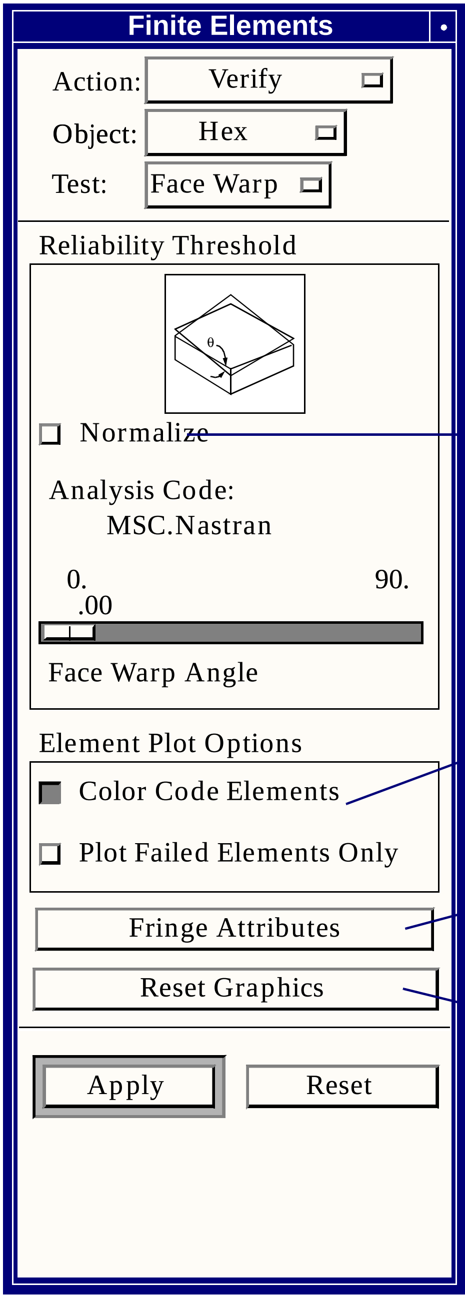
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Skew Angle is computed (p. 240)

Verify - Hex (Face Warp)

Each face of each hex element in the current group is tested for warp as if it were a quad. The warp angle is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its warp value in the command line. At completion, each element is color-coded according to the value computed for its warp value and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed warp angle will be divided by 15°. A "perfect" element will have no out-of-plane component and will therefore be the most reliable. When the Normalize button is turned OFF, the slidebar represents the computed Face Warp Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

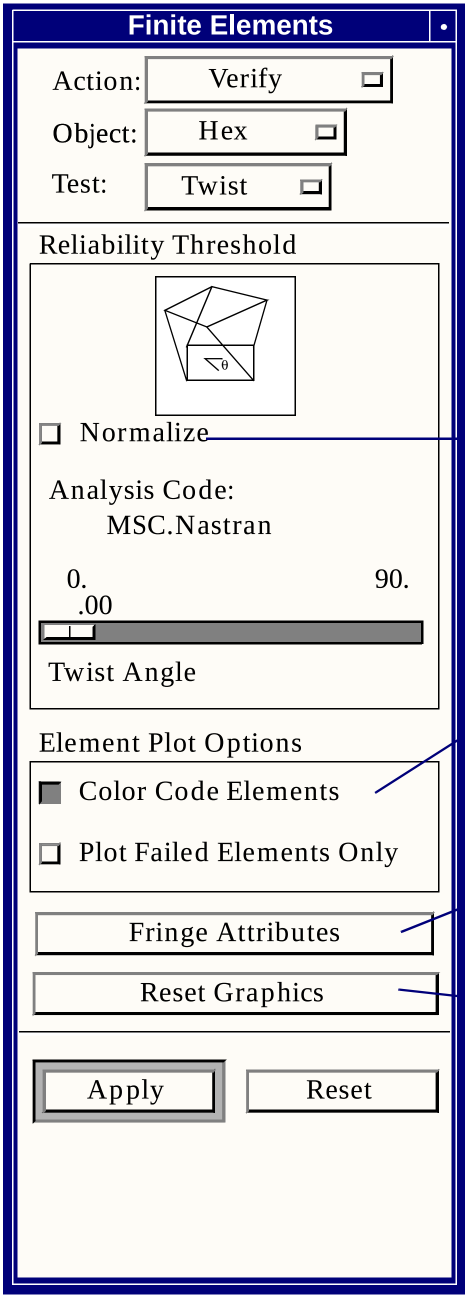
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Warp Angle is computed (p. 245)

Verify - Hex (Twist)

Each hex element in the current group is tested for maximum twist angle. Twist is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its maximum twist in the command line. At completion, each element is color-coded according to the value computed for its twist and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed twist angle will be divided by 90°. When the Normalize button is turned OFF, the slidebar represents the computed Twist Angle. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

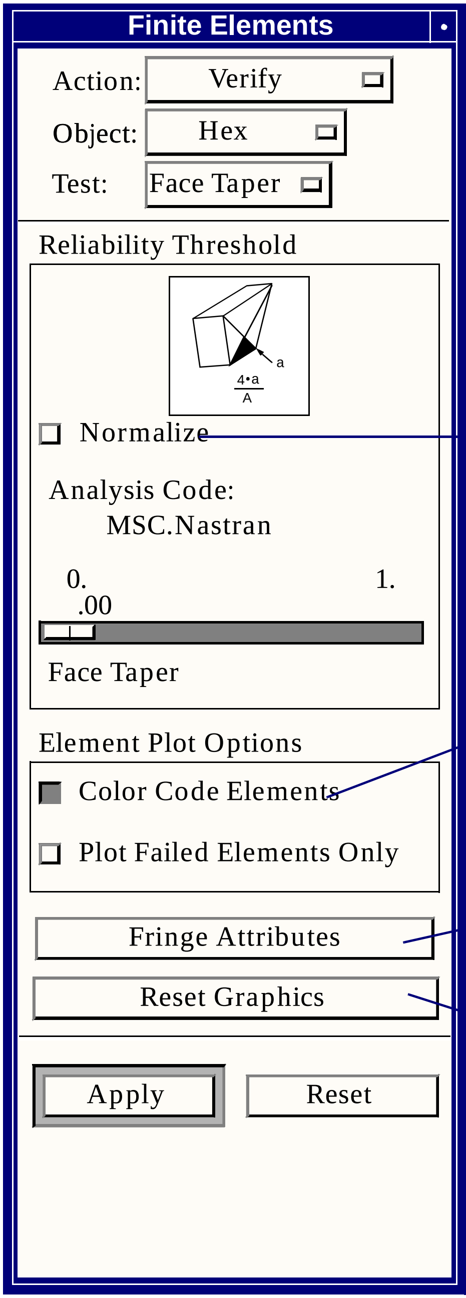
Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

More Help:

- How Twist is computed (p. 250)

Verify - Hex (Face Taper)

Each face of each hex element in the current group is tested for taper as if it were a quad element. Taper is obtained as described in **Theory** (p. 238). During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element’s ID number and its maximum taper in the command line. At completion, each element is color-coded according to the value computed for its taper and the current group is plotted in the Element Fill Render style.



When the Normalize button is selected, the computed taper ratio will be subtracted from 1. A “perfect” element face will have all 4 triangular subareas (a) equal, therefore 4*a/total area will equal 1 and the normalized equation will yield a Taper Factor of 0.

When the Normalize button is turned OFF, the slidebar represents the computed Taper Ratio. Move the slidebar to indicate the maximum acceptable value for your analysis type and code.

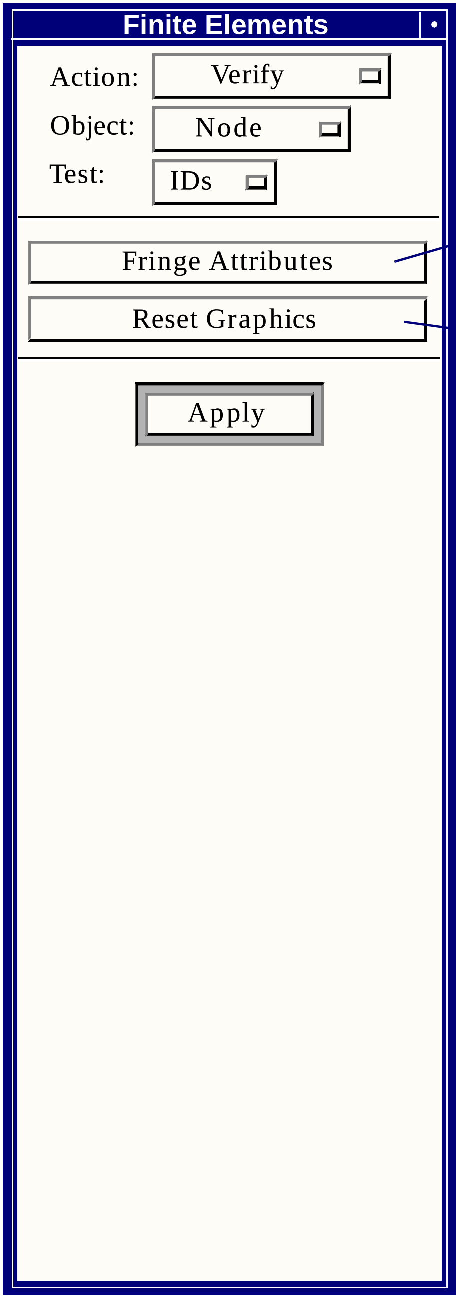
Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

Verify - Node (IDs)

Fringe plot based on node ID of nodes in the current group is generated. This display is helpful to check whether the node numbering has been optimized for bandwidth efficiency.

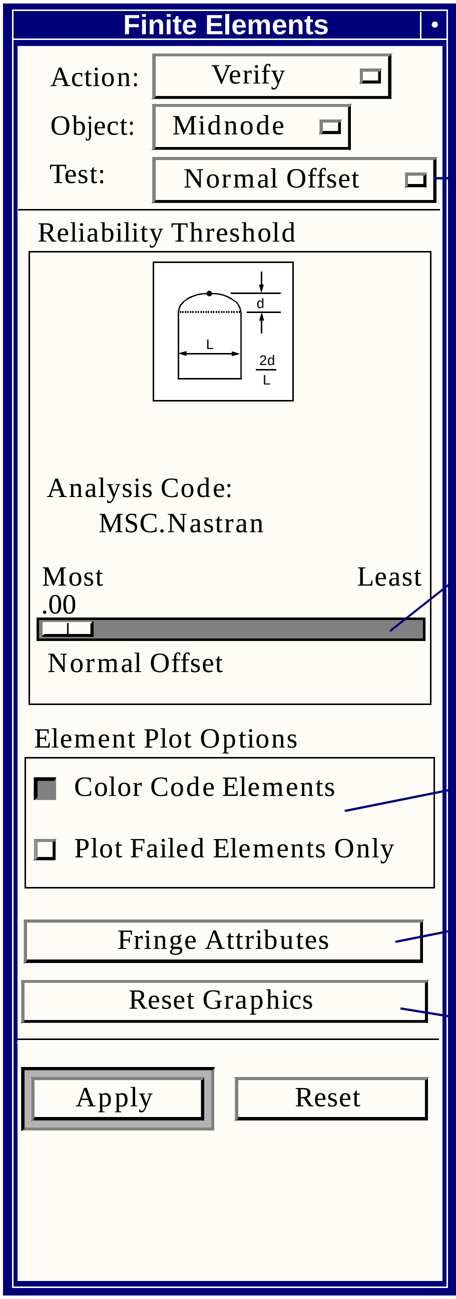


See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style.

Verify - Midnode (Normal Offset)

All quadratic order elements (those with mid-side nodes) in the current group are tested for deviation of the mid-side node from the mid-node position for an element with no curvature. The offset distance is measured perpendicular to a line that is the shortest distance between the corner nodes of that edge.



During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element's ID number and its Normal Offset value in the command line. At completion, each element is color-coded according its Normal Offset value. All elements exceeding the threshold value will be colored red. The current group is plotted in the Element Fill Render style.

Set the slidebar to an acceptable offset value for your analysis code. A "perfect" element will have the mid-side node directly at the mid-edge position, therefore, the offset d=0. The equation $\frac{2d}{L}$ will yield a value of zero and indicates the most reliable element for analysis.

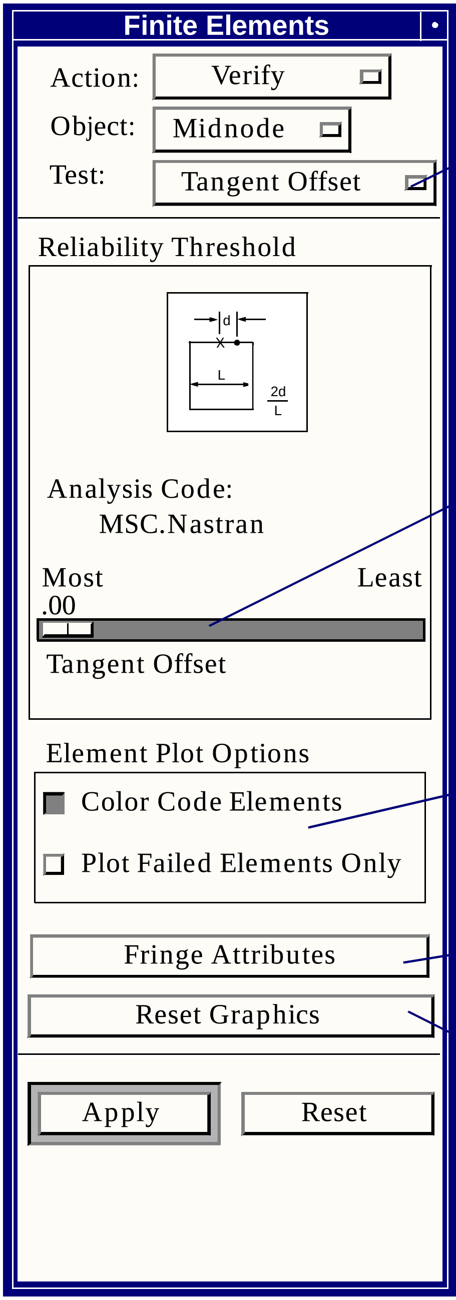
Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when the form was opened. This will usually turn OFF the Spectrum Display and change from Element Fill back to your original Render Style, and removing the vendor display from the viewport.

Verify - Midnode (Tangent Offset)

All quadratic order elements (those with mid-side nodes) in the current group in the active viewport are tested for deviation of the mid-side node from the mid-edge position of the element. The offset distance is measured along a line that is the shortest distance between the corner nodes of that edge.



During the check, MSC.Patran highlights any element exceeding the threshold value set by the slidebar, and echoes the element’s ID number and its Tangent Offset value in the Command Line. At completion, each element is color-coded according its Tangent Offset value. All elements exceeding the threshold value will be colored red. The current group is plotted in the Element Fill Render style.

Set the slidebar to an acceptable offset value for your analysis code. A “perfect” element will have the mid-side node directly at the mid-edge position; therefore, the offset d=0. The equation $\frac{2d}{L}$ will yield a value of zero and indicates the most reliable element for analysis.

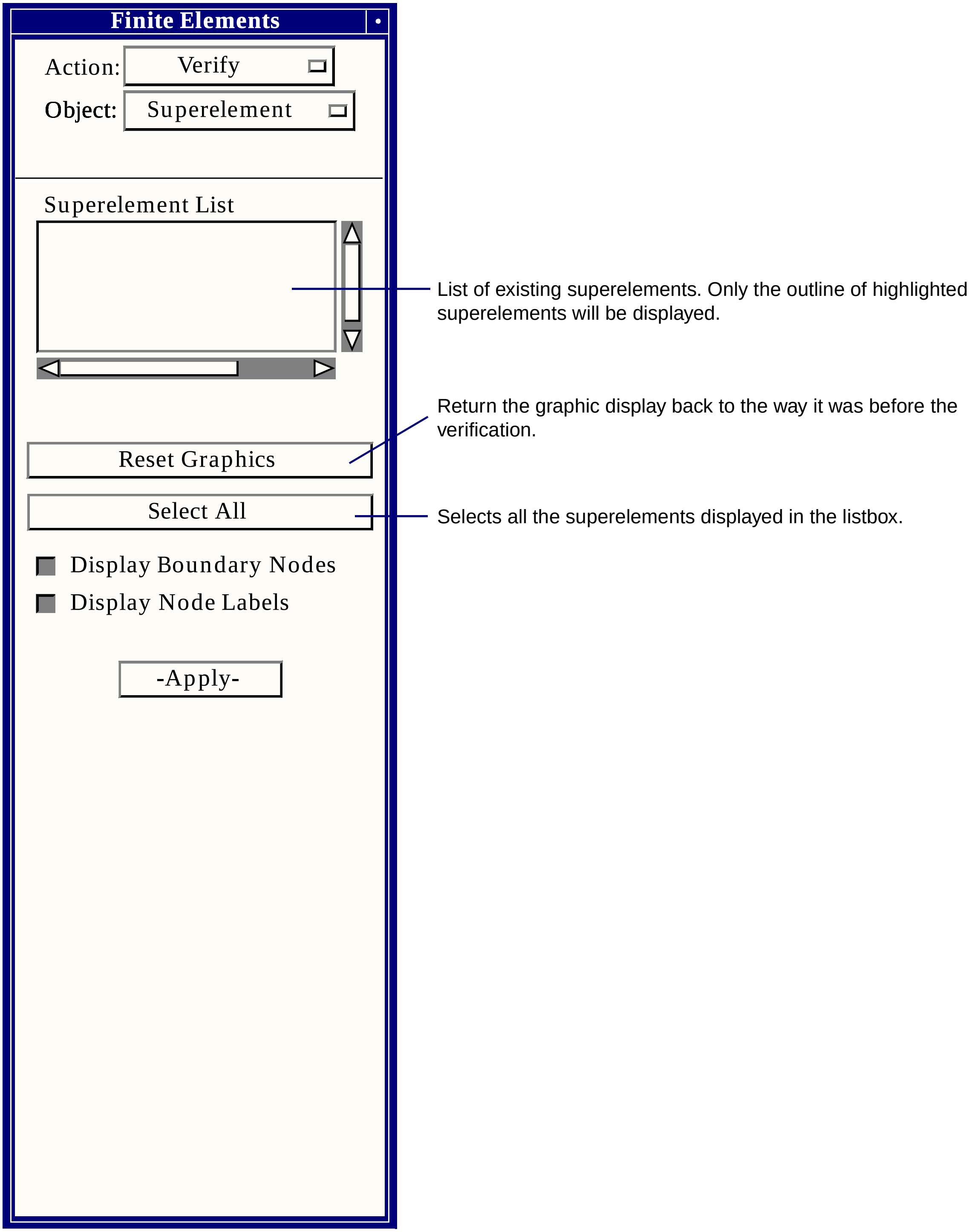
Toggle to control element plot options. You can either color code the elements and/or plot only the elements in the current group, which were tested and failed.

See **Fringe Attributes** (p. 247) in the *MSC.Patran Reference Manual, Part 6: Results Postprocessing*.

Returns your graphic display to the way it was when you entered the form. This will usually entail changing from Element Fill back to your original Render Style and removing the spectrum display from your viewport.

Superelement

This form verifies the selected superelements for any inconsistencies. Note that this is only available for the MSC.Nastran analysis preference.



11.3 Theory

Skew

Tria. Three potential skew angles are computed for each tria element. To calculate each skew angle, two vectors are constructed: one from a vertex to the mid-point of the opposite edge, and the other between the mid-points of the adjacent edges. The difference is taken of the angle between these two vectors and 90°. This procedure is repeated for the other two vertices. The largest of the three computed angles is reported as the skew angle for that element.

If Normalize is selected on the verification form, the skew angle is divided by 90° to yield the skew factor. An equilateral triangle will have a skew factor of 0.

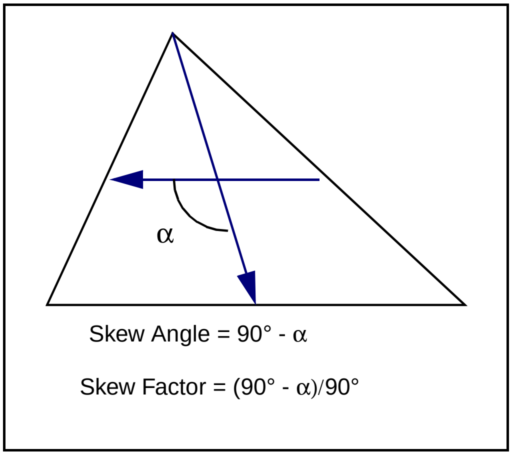


Figure 11-2 Tria Skew Angle

Quad. Prior to testing for skew, each element is first checked for convexity. Elements which fail the convexity check “double back” on themselves causing their element stiffness terms to have either a zero or negative value.

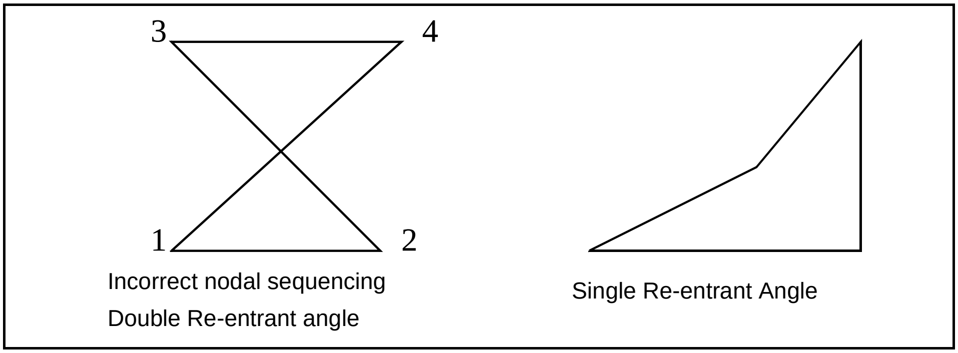


Figure 11-3 Convexity Check

This skew test is based on a reference frame created by first bisecting the four element edges, creating an origin at the vector average of the corners, where the x-axis extends from the origin to the bisector on edge 2. The z-axis is in the direction of the cross product of the x-axis and the vector from the origin to the bisector of edge 3. The y-axis is in the direction of the cross product of the x and z axis as shown in **Figure 11-4**.

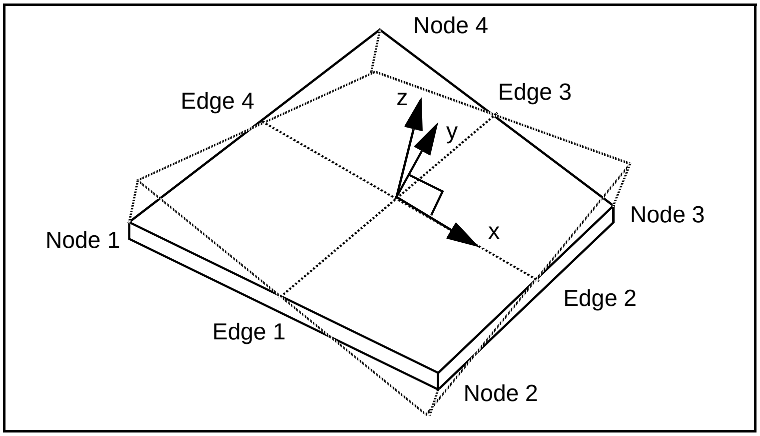


Figure 11-4 The Element Test Coordinate System

The Robinson and Haggenmacher[1] skew test uses the angle alpha between the edge 2 and 4 bisector and the test y-axis. The resulting angle is subtracted from 90° to yield the skew angle.

If Normalize is selected on the verification form, the skew angle is divided by 90° to yield the skew factor. A square element will have a skew factor of 0.

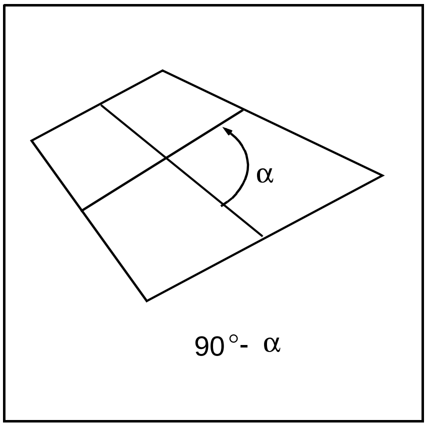


Figure 11-5 Quad Skew Angle

Tet. Each face of the tet element is tested for skew as if it were a tria element. See explanation for computation of skew angle - **Tria** (p. 238). The highest resulting angle for each element is retained as the skew angle.

[1] J. Robinson and G. W. Haggenmacher, “Element Warning Diagnostics,” *Finite Element News*, June and August, 1982.

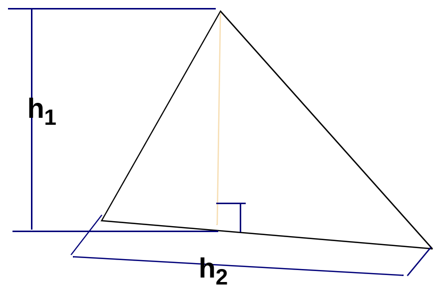
Wedge. Each face of the wedge element is tested for skew as if it were either a quad or tria element. See explanation for computation of skew angle - **Tria** (p. 238) or **Quad** (p. 238). The highest resulting angle for each element is retained as the skew angle.

Hex. Each face of the hex element is tested for skew as if it were a quad element. See explanation for computation of skew angle - **Quad** (p. 238). The highest resulting angle for each element is retained as the skew angle.

Aspect Ratio

Tria. The aspect ratio for a triangle is calculated as the ratio of the length, h2, of the edge of a triangle, to the height, h1. The ratio of h2 to h1 is then multiplied by $\sqrt{3}/2$ such that a "perfect" element in the shape of an equilateral triangle will equal one. This procedure is repeated for the remaining two edges of the triangle, and the largest value is retained as the aspect ratio for the element.

If Normalize is selected on the verification form, then the aspect ratio is inverted such that it becomes less than or equal to one. This inverted aspect ratio is subtracted from one to yield the aspect factor. An equilateral triangle will have an aspect factor of 0.



Aspect Ratio = $\dfrac{\sqrt{3}h_2}{2h_1}$ Aspect Factor = $-\dfrac{2h_1}{\sqrt{3}h_2}$

Figure 11-6 Tria Aspect Ratio

Quad. The aspect ratio for a quad is derived from one test proposed by Robinson and Haggenmacher[1]. This test is based on projection plane created by first bisecting the four element edges, creating a point on the plane at the vector average of the corners. The x-axis extends from the point to the bisector on edge 2. The ratio is determined as the ratio of the length from the origin to the bisector of edge 2 and the length from the origin to the bisector of edge 3. If the ratio is less than 1.0, it is inverted.

[1]J. Robinson and G. W. Haggenmacher, "Element Warning Diagnostics," *Finite Element News*, June and August, 1982.

If Normalize is selected on the verification form, then the aspect ratio is inverted such that it becomes less than or equal to one. This inverted aspect ratio is subtracted from one to yield the normalized aspect ratio. A square element will have a normalized aspect ratio of 0.

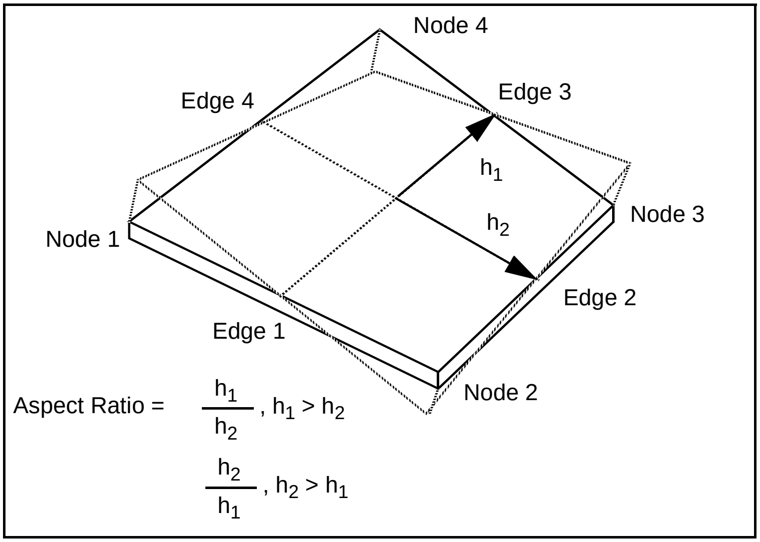


Figure 11-7 Quad Aspect Ratio

Tet. The aspect ratio for a tet element is computed by taking the ratio of the height of a vertex to the square root of the area of the opposing face. This value is then manipulated in one of two ways, depending on whether the Normalize parameter is selected on the verification form.

If Normalize is NOT selected, the maximum height to area value is multiplied by a factor $C = 0.805927$, which is the ratio of height to edge length for an equilateral tetrahedron. This result is reported as the Aspect Ratio. An equilateral tet will report a value of 1.
Aspect Ratio = $Max(C_f \cdot h_i / \sqrt{A_i})$, i = 1, 2, 3, 4 .

If Normalize IS selected, the maximum height to area value is inverted and subtracted from 1.
Aspect Factor = $(1 - 1/(MaxC_f \cdot h_i / \sqrt{A_i}))$, i = 1, 2, 3, 4 .

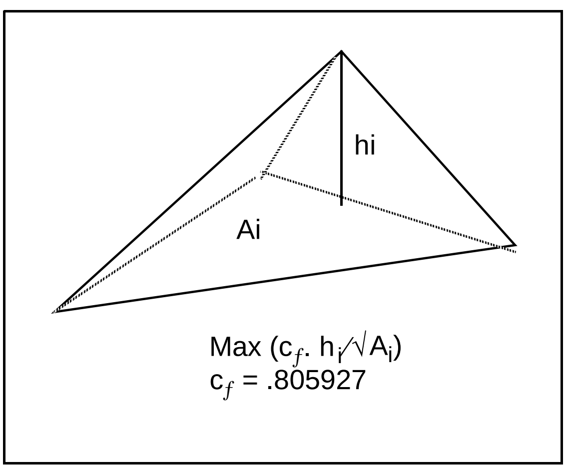


Figure 11-8 Tet Aspect Ratio

Wedge. MSC.Patran averages the two triangular faces of the wedge element to obtain a mid-surface. The aspect ratio of this triangular mid-surface is computed $(\sqrt{3}h_2/2h_1)$. Next the height (h_1) of the wedge is compared to the maximum edge length of the mid-surface (h_4).

If the height of the wedge is greater than the maximum edge length then the aspect ratio for the wedge element equals the mid-surface aspect ratio multiplied by the maximum edge length divided by the distance between the triangular faces (h_3).

If the height of the wedge is less than the maximum edge length then the aspect ratio for the wedge element equals either the mid-surface aspect ratio or the maximum edge length divided by the distance between the triangular faces, whichever is greater.

If Normalize is selected on the verification form, then the aspect ratio is inverted such that it becomes less than or equal to one. This inverted aspect ratio is subtracted from one to yield the aspect factor. An equilateral wedge element will have an aspect factor of 0.

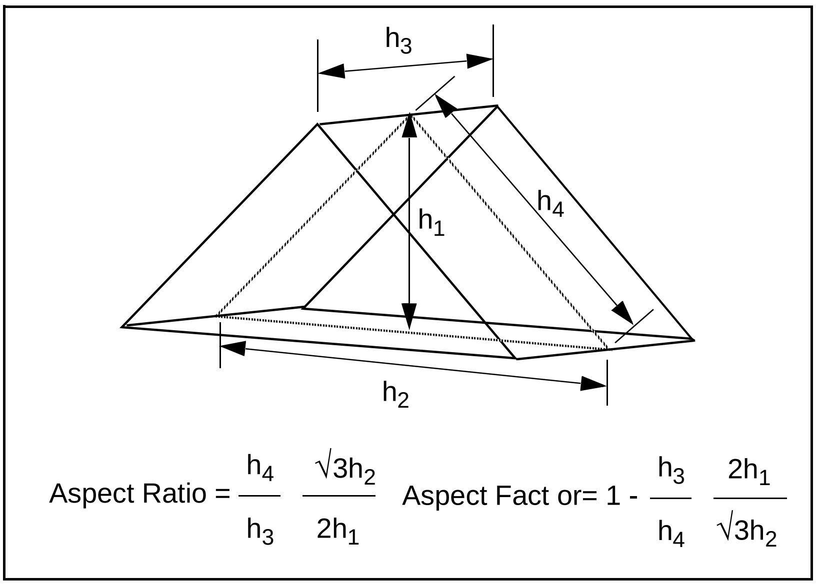
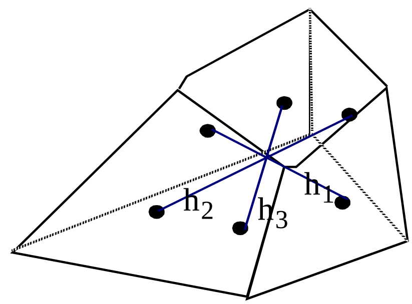


Figure 11-9 Wedge Aspect Ratio

Hex. The aspect ratio is calculated as the ratio of the distance between opposing faces. This distance is determined by treating each HEX face as if it were a warped quadrilateral. Each face is processed to produce a projected plane. The distances between the centerpoints of all three pairs of opposing faces are compared.The aspect ratio is determined by taking the maximum distance between any two faces and dividing it by the minimum distance between any two faces.

If Normalize is selected on the verification form, then the aspect ratio is inverted such that it becomes less than or equal to one. This inverted aspect ratio is subtracted from one to yield the aspect factor. A cubic element will have an aspect factor of 0.



$$\text{AspectRatio} = \frac{\max(h_1, h_2, h_3)}{\min(h_1, h_2, h_3)} \qquad \text{AspectFactor} = 1 - \frac{\min(h_1, h_2, h_3)}{\max(h_1, h_2, h_3)}$$

Figure 11-10 Hex Aspect Ratio

Warp

Quad. The warp test is a test proposed by Robinson and Haggenmacher[1] which uses the following method of calculating the Quad element Warp. This test is based on a projection plane created by first bisecting the four element edges, creating a point on the plane at the vector average of the corners, where the x-axis extends from the point to the bisector on edge 2. The plane normal is in the direction of the cross product of the x-axis and the vector from the origin to the bisector of edge 3. Every corner of the quad will then be a distance "h" from the plane. The length of each half edge is measured and the shortest length is assigned "l." The warp angle is the arcsine of the ratio of the projection height "h" to the half edge length "l."

If Normalize is selected on the verification form, the warp angle is divided by 15° to yield the warp factor. A planar element has a warp factor of 0.

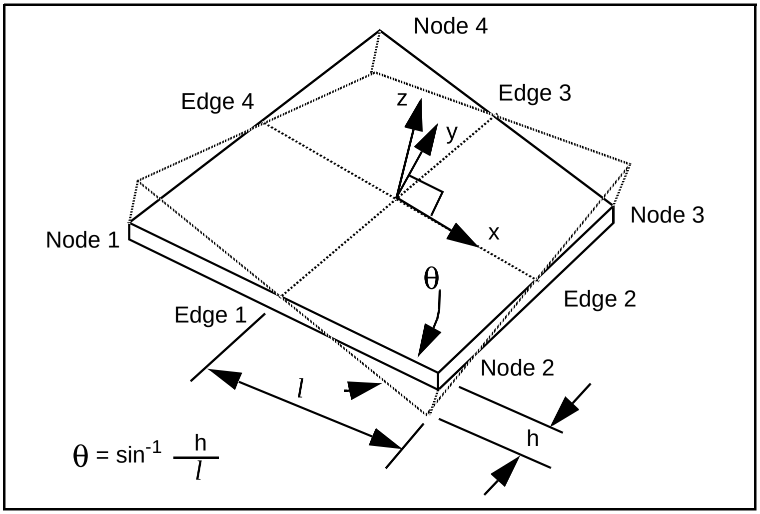


Figure 11-11 Quad Warp Angle

Wedge. Each quad face of the wedge element is tested for warp as if it were a quad element. See explanation for computation of warp angle - **Quad** (p. 245). The highest resulting angle for each element is retained as the warp angle.

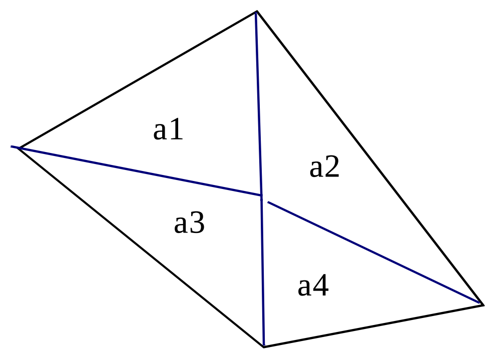
Hex. Each face of the hex element is tested for warp as if it were a quad element. See explanation for computation of warp angle - **Quad** (p. 245). The highest resulting angle for each element is retained as the warp angle.

[1]J. Robinson and G. W. Haggenmacher, "Element Warning Diagnostics," Finite Element News, June and August, 1982.

Taper

Quad. The taper test is a test proposed by Robinson and Haggenmacher[1] which uses the following method of calculating the Quad element. Taper four triangles are created bounded by the element edge and the edges created by connecting the element verification reference frame origin with the two nodes at the element edge. The resulting four triangular areas are calculated and then summed. The ratio of the area with the smallest triangle and the total area of the element is taken as the taper ratio.

If Normalize is selected on the verification form, the taper ratio is subtracted from one to yield the taper factor. A square element has a taper factor of 0.



$$\text{Taper Ratio} = \frac{4a_{smallest}}{1 + a2 + a3 + a4}$$

Figure 11-12 Quad Taper

Wedge. Each quad face of the wedge element is tested for taper as if it were a quad element. See explanation for computation of taper - **Quad** (p. 246). The lowest resulting value for each element is retained as the value of face taper.

Hex. Each face of the hex element is tested for taper as if it were a quad element. See explanation for computation of taper - **Quad** (p. 246). The lowest resulting value for each element is retained as the value of face taper.

Edge Angle

Tet. Edge angle measures the angle between adjacent faces of the tetrahedral element. In an equilateral tetrahedral element, this angle will equal 70.259°. The largest angle found in the element is retained. MSC.Patran then computes the absolute value of the difference between the measured angle and 70.259°. This is the value reported as the Edge Angle.

If Normalize is selected on the verification form, the edge angle is divided by 110° to yield the edge angle factor. An equilateral tet will have an edge angle factor of 0.

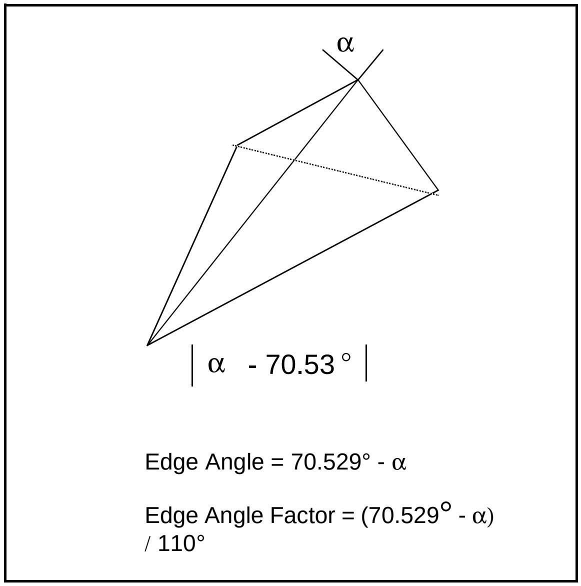


Figure 11-13 Tet Edge Angle

Wedge. An edge angle is the absolute value of the angle between the two faces meeting at an edge subtracted from the ideal angle for that edge. The ideal angle between two quad faces is 60 degrees, and the ideal angle between a quad face and a tria face is 90 degrees. For warped quad faces, the projected plane of the face is used to compute the face normal used in the angle calculation. The maximum edge angle is calculated for each wedge element.

If Normalize is selected on the verification form, the edge angle is divided by 60° to yield the edge angle factor.

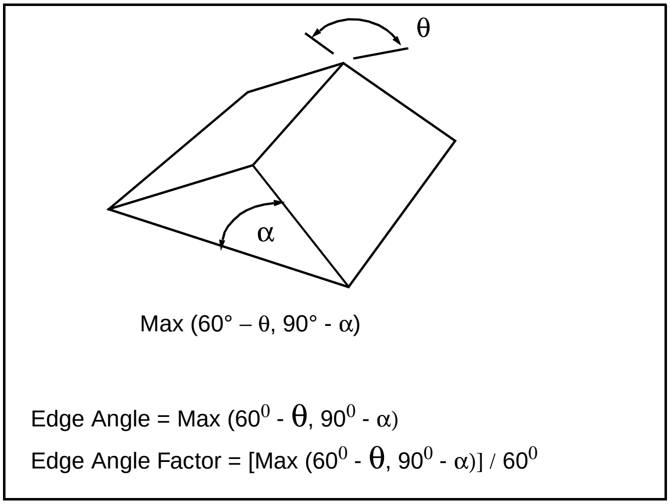


Figure 11-14 Wedge Edge Angle

Hex. An edge angle is the absolute value of the angle between the two faces meeting at an edge subtracted from the ideal angle for that edge. The ideal angle between faces of a hex element is 90 °. For warped faces, the projected planes for each face is used to compute the face normals used in the angle calculation. The maximum edge angle is calculated for each hex element.

If Normalize is selected on the verification form, the edge angle is divided by 90° to yield the edge angle factor.

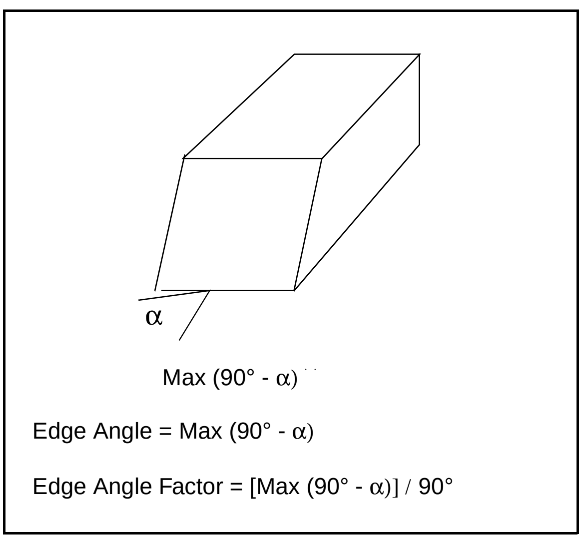


Figure 11-15 Hex Edge Angle

Collapse

Tet. Collapse is an indicator of near zero volume tetrahedral elements. The test takes the ratio of the height of a vertex to the square root of the area of the opposing face. This value approaches zero as the volume of the element approaches zero.

If Normalize is NOT selected on the verification form, the minimum height to area value is multiplied by a factor $C_f = 0.805927$, which is the ratio of height to edge length for an equilateral tetrahedron. An equilateral tet will report a value of 1. Collapse $= \left(\mathrm{Min} \frac{C_f \times h_i}{\sqrt{A_i}} \right), i = 1, 2, 3, 4$.

If Normalize IS selected, the minimum height to area value is subtracted from 1. An equilateral tet will report a value of 0. Collapse Factor $= \quad - \mathrm{Min}\left(\frac{C_f \times h_i}{\sqrt{A_i}} \right)$, i = 1,2,3,4.

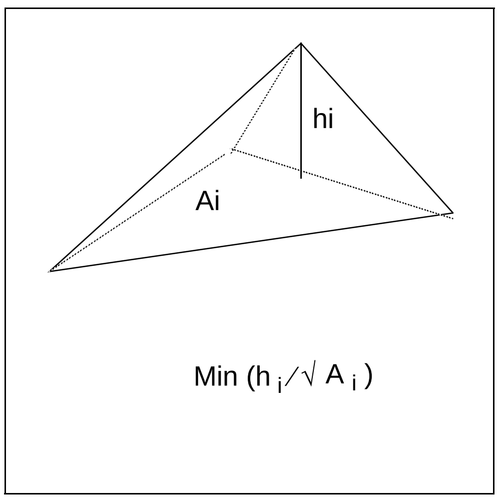


Figure 11-16 Tet Collapse

Twist

Wedge. Twist is the rotation of one face of a solid with respect to its opposite face. To compute twist angle, normals are drawn from the center of each tria surface. These vectors are projected onto a plane. The angular difference between the two vectors is the twist angle.

If Normalize is selected on the verification form, the twist angle is divided by 60° to yield the twist factor.

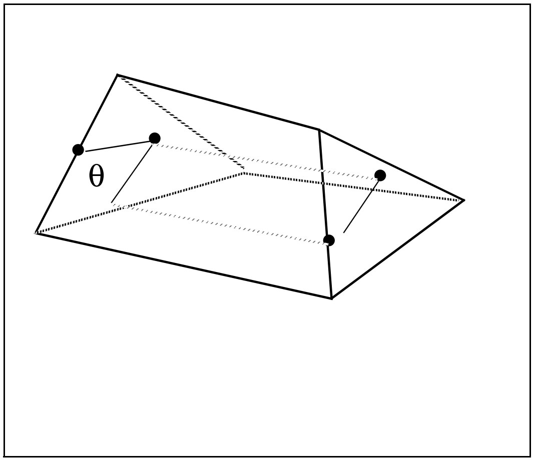


Figure 11-17 Wedge Twist Angle

Hex. Twist is the rotation of one face of a solid with respect to its opposite face. A twist angle is computed about all three principal axes of hex elements. To compute the twist angle, each face is treated as if it were a warped quad. Vectors from the center of the projected plane to the middle of two adjacent edges are constructed. The vectors are summed to compute a reference vector. The same steps are performed for the opposite face. A line through the center of each projected face and the plane normal to this line is determined. The two reference vectors are projected onto this plane and the angular difference between them is measured. The highest angle found is retained as the twist angle.

If Normalize is selected on the verification form, the twist angle is divided by 90° to yield the twist factor.

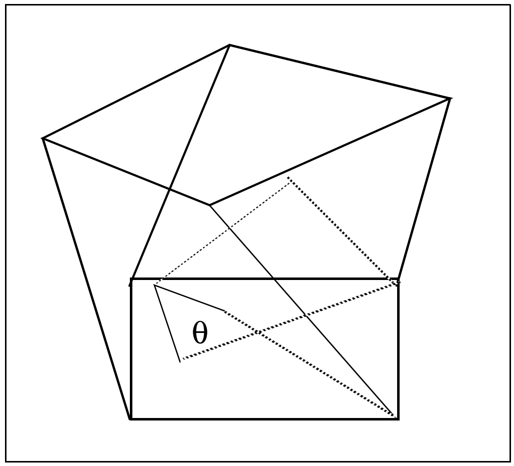


Figure 11-18 Hex Twist Angle

CHAPTER
12 The Show Action

- Show Forms

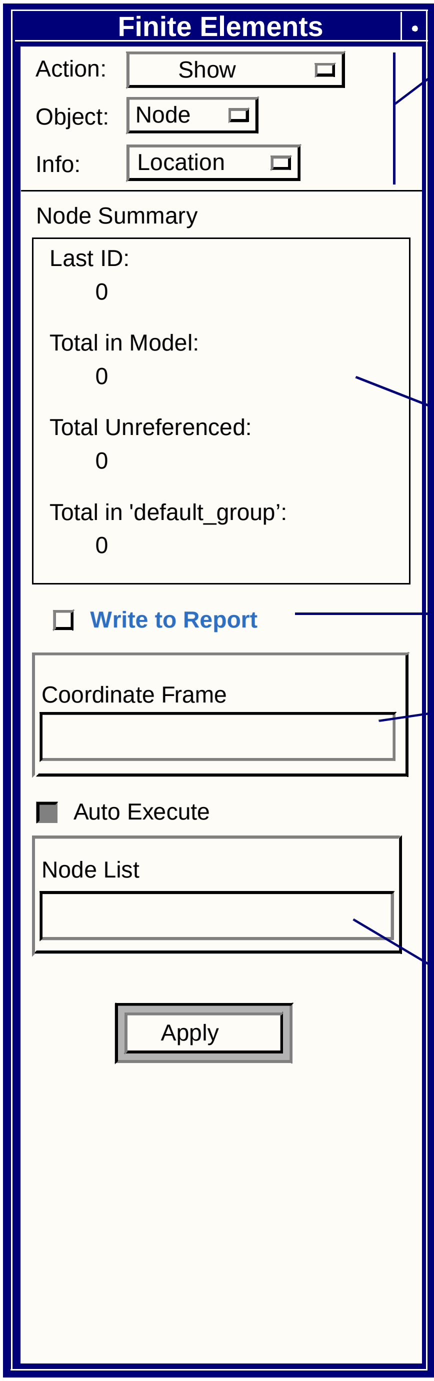
12.1 Show Forms

When *Show* is the selected Action, the following options are available.

Object	Info	Description
Node	**Location**	Displays the location of the selected nodes in a selected coordinate system or the reference coordinate system in which the node was created, the reference coordinate system. The reference coordinate system ID and the analysis coordinate system ID are also displayed.
	Distance	Displays the straight-line distance between the nodes in the first-node list and the second-node list.
Element	**Attributes**	Displays the element ID, topology (i.e., element type), parent geometry, number of nodes in the element, load and boundary conditions, material property ID number, element properties and results associated with the selected elements.
	Coordinate System	Plots the element coordinate systems of the selected elements.
Mesh Seed	**Attributes**	
Mesh Control	**Attributes**	
MPC		Displays Multi-Point Constraint (MPC) type and information about the associated constraint equation terms for selected MPCs.
Connector	**Attributes**	Displays the Connector attributes.

Show - Node Location

When *Show* is the selected Action and *Node* is the selected Object, the Show Node menu is displayed. This is used to view either the location of or distance between selected nodes.



The Show Node Location menu displays the location of the selected nodes in the reference coordinate system, giving the reference coordinate system ID, and the analysis coordinate system ID.

More detailed information than what is provided in the resulting information table can be obtained by selecting the individual cells of the **The Show Action Information Form** (p. 569) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*. This additional information is then displayed in the textbox directly below the table.

The Node Summary table lists information about the existing nodes in the current database, including the highest node ID currently used, the total number of nodes in the model, the number of unreferenced nodes, and the total number of nodes in the current group.

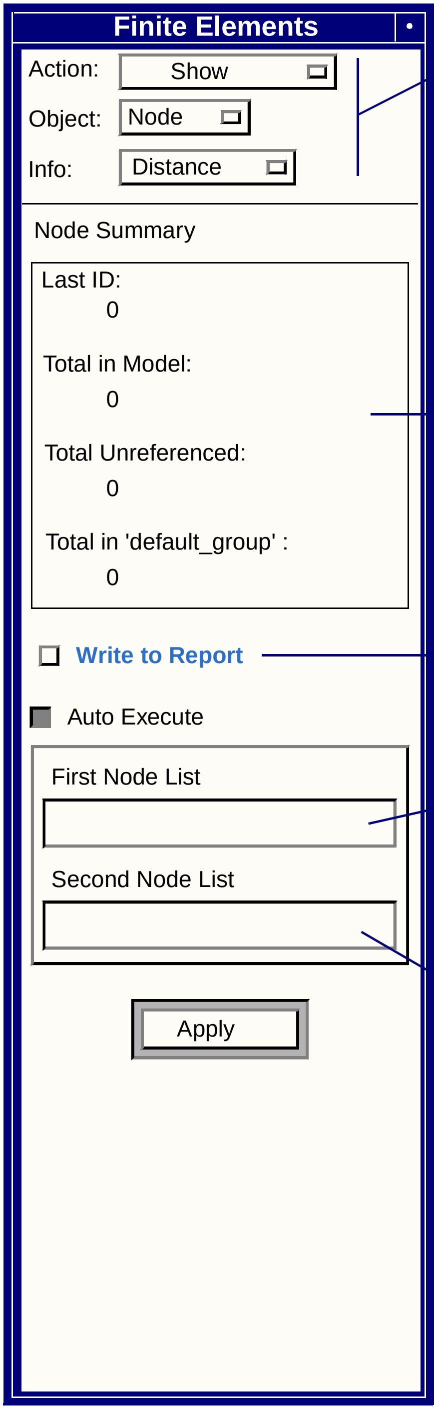
See **Write to Report** (p. 258)

Used to input the coordinate frame in which the coordinate values of the selected nodes are to be shown. Coordinate frames can be specified by selecting this box and then cursor selecting, or by typing “Coord n” where n is the ID of the desired coordinate frame. If no coordinate frame is specified, the nodes will be shown in the coordinate frame in which they were created.

The Node List textbox is where the list of nodes to be shown is specified. Nodes can be specified by selecting this box and then selecting the nodes with the cursor, or by typing in “node n” where n is the number of the desired node.

Show - Node Distance

When *Show* is the selected Action and *Node* is the selected Object, the Show Node menu is displayed. This is used to view either the location of or distance between selected nodes.



The Show Node Distance menu displays the straight-line distance between the nodes in the first-node list and the second-node list.

More detailed information than what is provided in the resulting information table can be obtained by selecting the individual cells of the **The Show Action Information Form** (p. 569) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*. This additional information is then displayed in the textbox directly below the table.

The Node Summary table lists information about the existing nodes in the current database, including the highest node ID currently used, the total number of nodes in the model, the number of unreferenced nodes and the total number of nodes in the current group.

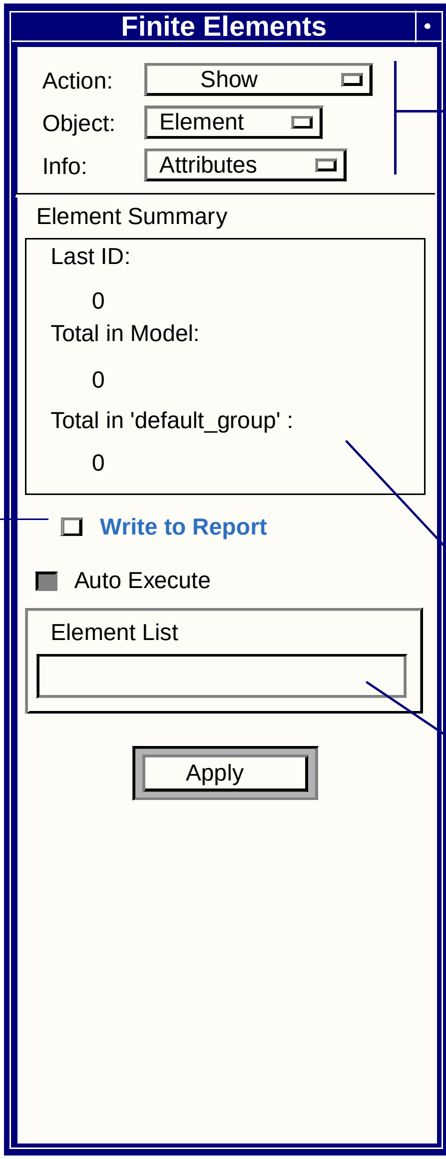
See **Write to Report** (p. 258).

The First Node List textbox is used to input the list of nodes where the distance measurement starts. Nodes can be specified by selecting this box and then selecting the nodes with the cursor, or by typing in “node n” where n is the number of the desired node.

The Second Node List textbox is used to input the list of nodes where the distance measurement ends. If the lists do not contain the same number of points, the last point of the shorter list will be repeated until the longer list is exhausted.

Show - Element Attributes

When *Show* is the selected Action, *Element* is the selected Object and Attributes is the selected Info, the Show Element Attributes menu is displayed. This is used to display the element attributes of selected elements.



The Show/Element/Attributes menu displays the element ID, topology (i.e., element type), parent geometry, number of nodes, loads and boundary conditions, material property ID number, element properties and results associated with selected elements in an information table.

More detailed information than what is provided in the resulting information table can be obtained by selecting the individual cells of the **The Show Action Information Form** (p. 569) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*. This additional information is then displayed in the textbox directly below the table.

Selecting this menu automatically brings up the element select menu which allows the user to select elements from the graphics window via cursor pick or element type.

The Element Summary table lists information about the existing elements in the current database including the highest ID number currently used, the total number of elements in the model, and the total number of elements in the current group.

The Element List textbox is where the list of elements to be shown is specified. Elements can be specified by using the mouse button on this box, and then selecting the elements with the cursor, or by typing in “element n” where n is the number of the desired element. Elements can also be selected by type through the Element select menu.

See **Write to Report** (p. 258)

Write to Report

When toggled ON, the **File>Report** (p. 212) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* will appear. If the user proceeds to write attributes within the Report File form, the user will have information for all the entities in the database. Note: This can be done without selecting entities in the Finite Elements form.

Set and keep a file in an open state for subsequent output from the Finite Element form. In order to output information for selected entities (a subset of the database) to a file, perform the following:

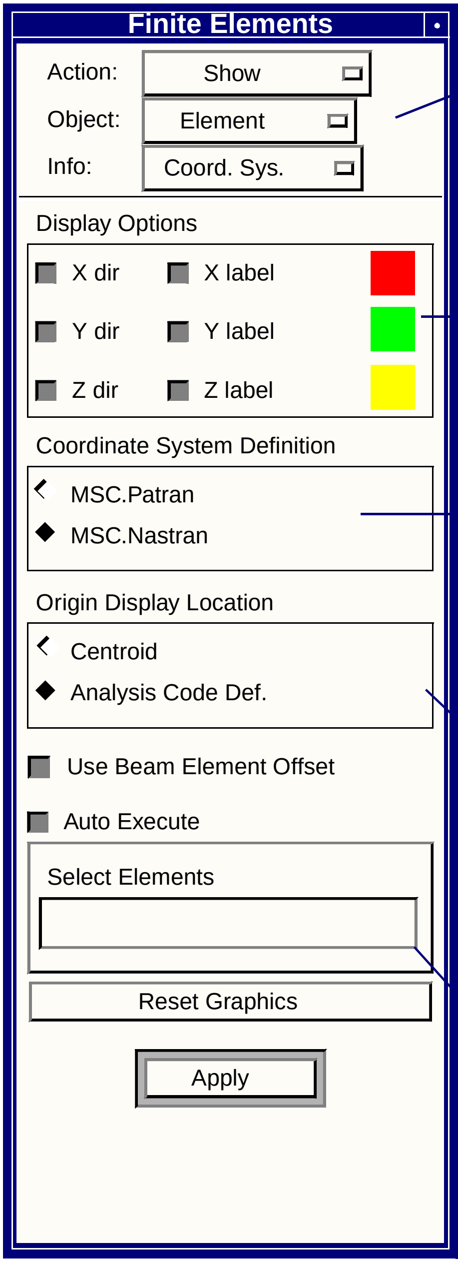
1. On the Finite Element form, toggle ON the Write To Report toggle. The Report File form will appear.
2. On the Report File form, set the Output Format, File Width and Open File.
3. On the Report file form, select an existing report file or create a new one.

Important:	Do not click Apply (button located on the lower right of the Report file form). This will immediately dump all the database entities to the file.

4. Click Cancel to hide the Report file form.
5. Proceed to select the desired entities and generate an information spreadsheet. This will also write the same information to the output text file.

Show - Element Coordinate System

When *Show* is the selected Action, *Element* is the selected Object and Coord. Sys. is the selected Info, the Show Element Coord. Sys. menu is displayed. This is used to plot the element coordinate systems for the selected elements.



The Show/Element/Coord. Sys. menu plots the element coordinate systems.

Important: Selecting this menu automatically brings up the element select menu which allows the user to select elements from the graphics window via cursor pick or element type.

The Display Options section is used for selecting which element coordinate system axis will be displayed (X dir, Y dir, Z dir toggles), which axis labels will be displayed (X label, Y label, Z label toggles) and the color for each axis.

The Coordinate System Definition can be set to :

MSC.Patran for displaying MSC.Patran's definition of the element coordinate system.

MSC.Nastran for displaying MSC.Nastran's definition of the element coordinate system.

The Origin Display Location can be set to :

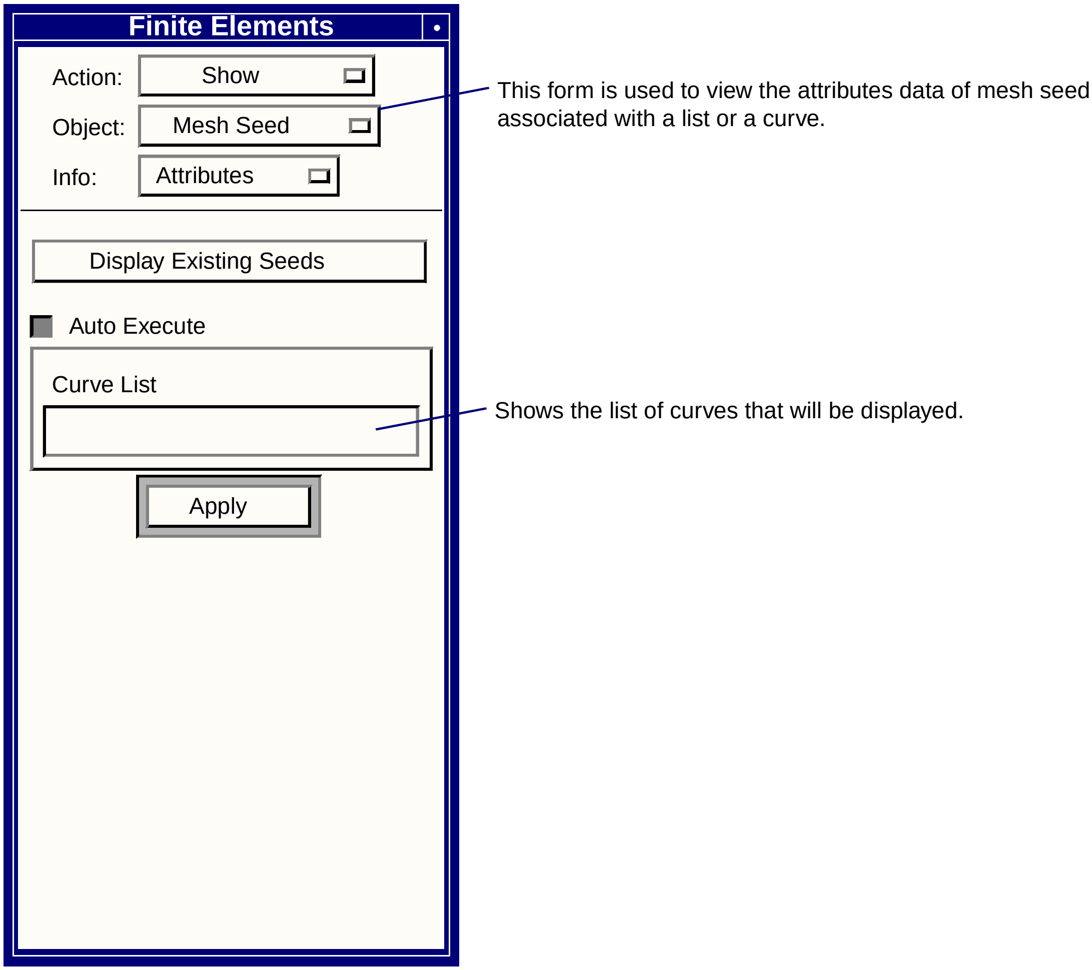
Centroid to display the element coordinate system origin at the centroid of the element (same as MSC.Patran's coordinate system definition).

Analysis Code Def. to display the element coordinate system origin at the selected coordinate system definition of the origin.

Return your graphic display to the way it was when you entered the form.

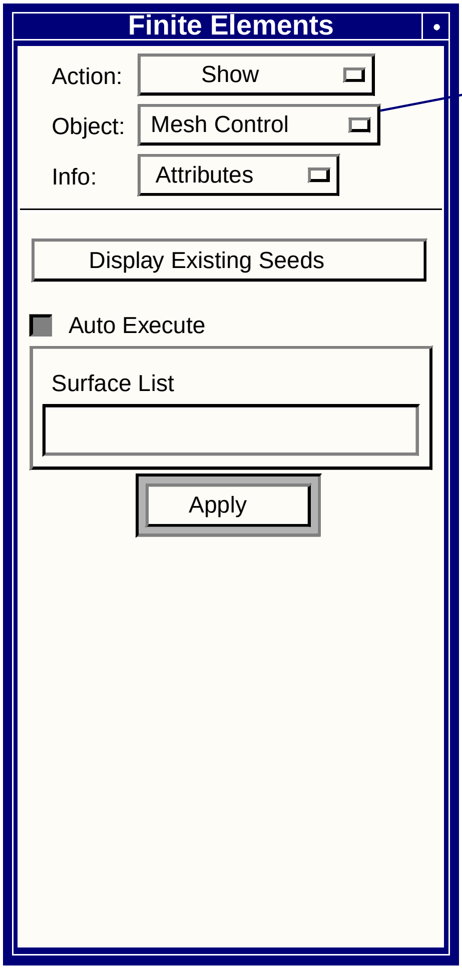
Show - Mesh Seed Attributes

When *Show* is the selected Action, *Mesh Seed* is the selected Object and Attributes is the selected Info, the Show Mesh Seed Attributes menu is displayed. This is used to show the mesh seed attributes for the selected curve.



Show - Mesh Control Attributes

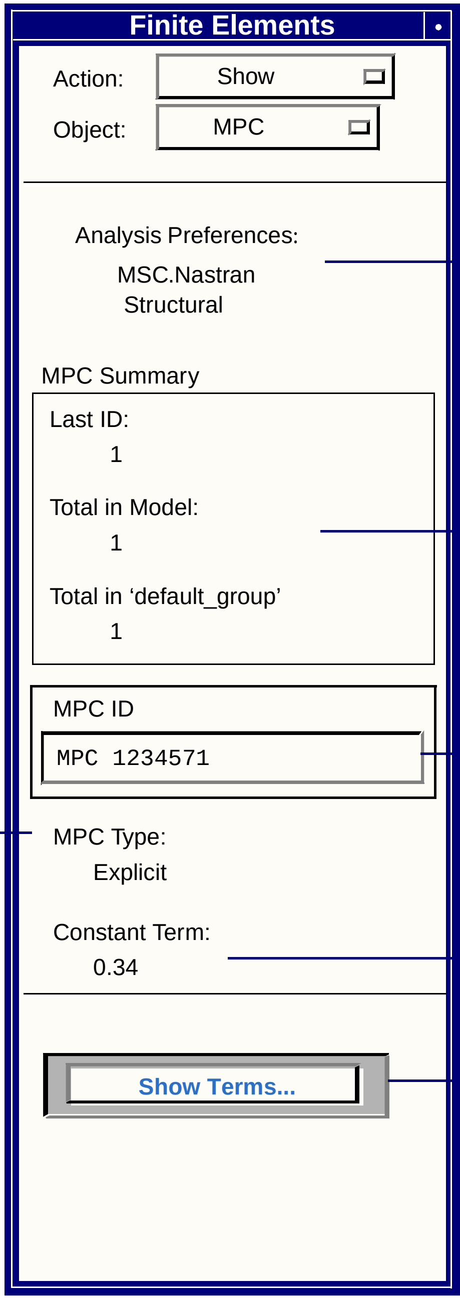
When *Show* is the selected Action, *Mesh Control* is the selected Object and Attributes is the selected Info, the Show Mesh Control Attributes menu is displayed. This is used to show the mesh control attributes for the selected surfaces.



This form is used to view the attributes data of mesh control on a list of surfaces

Show - MPC

When *Show* is the selected Action and *MPC* is the selected Object, the Show MPC form is displayed. Use this to view the attributes of existing MPCs.



Indicates the current settings of the Analysis Code and Analysis Type Preferences.

The MPC Summary table lists statistics about the existing MPCs in the database, including the highest ID currently used, the total number of MPCs in the database and the total number of MPCs in the current group.

Used to specify the MPC to be shown. The form will be updated automatically whenever the contents of this databox are changed to reflect the attributes of the existing MPC. If the MPC does not exist, the remaining widgets in this form are not displayed.

Indicates the value of the Constant Term, if supported by this MPC type.

Brings up the Show Terms form. This form is used to view the dependent and independent terms.

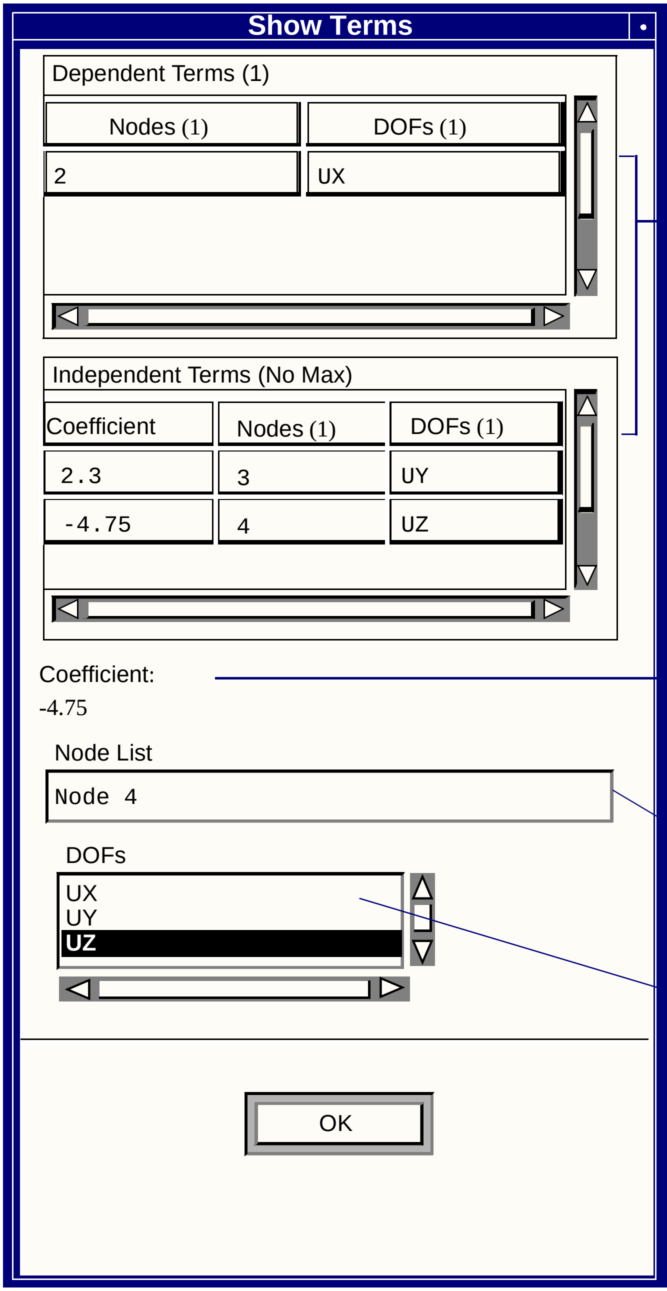
Indicates the MPC type currently selected. If the MPC is not valid for the current Analysis Code and Analysis Type preferences, this label will read:

Not Valid for Current Preferences

In this case, the remaining widgets will not be displayed.

Show - MPC Terms

The *Show Terms* form appears when the Show Terms button is selected on the Show MPC menu. Use this form to view information about the *dependent* and *independent* terms of an MPC.



Holds the dependent and independent term information as rows in the spreadsheet. A term consists of one or more of the following:

1) A sequence number (not shown)

2) A nonzero coefficient

3) A list of nodes

4) A list of degrees-of-freedom

Terms from these spreadsheets can be selected for an expanded listing of term components.

A *Sequence Label* (not shown) indicating the value of the sequence of the selected term is displayed only if a term including a sequence is selected.

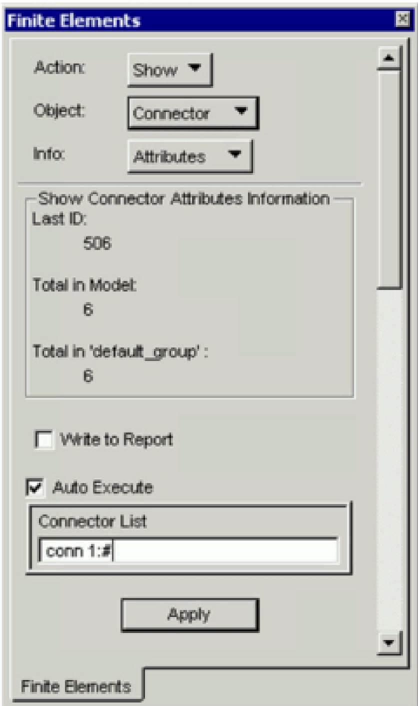
Coefficient Label indicating the value of the coefficient of the selected term, which is displayed only if a term including a coefficient is selected.

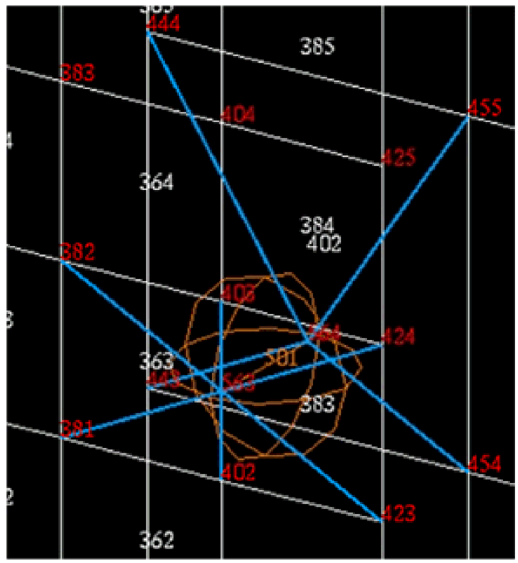
Node List textbox indicating the nodes associated to the selected term, which is displayed only if a term is selected.

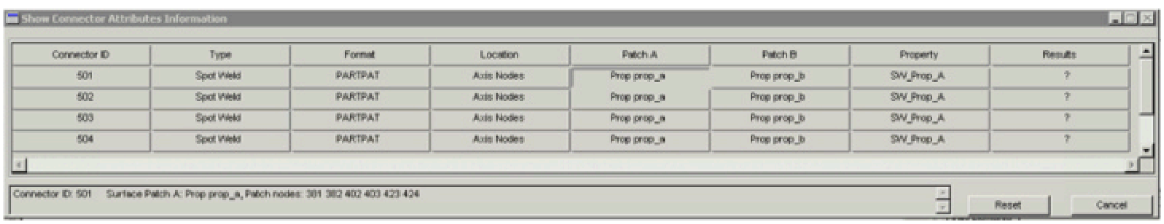
DOF listbox (read only) indicating the degrees-of-freedom associated with the selected term, which is displayed only if a term including degrees-of-freedom is selected.

Show Connectors

The form under the Show action shall simply present a Connector select databox (with an auto-exec toggle, on by default), allowing the user to select as many connectors as he/she wishes. Upon selection, a spreadsheet form shall be presented to show the values of each attribute of the selected connectors. Wherever appropriate, if a cell for an attribute is selected, then whatever additional information that is available for that attribute is presented in a text window below the spreadsheet, as is standard for most spreadsheets in Patran. For example, if the connector property name is selected, then the attributes of that connector property are shown.






Show Connector Attributes Information

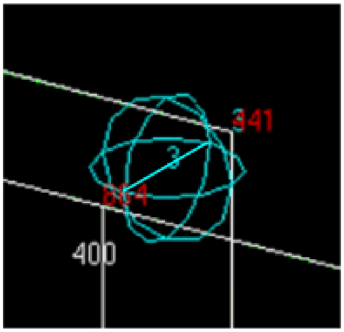
Connector ID	Type	Format	Location	Patch A	Patch B	Property	Results
501	Spot Weld	PARTPAT	Axis Nodes	Prop prop_a	Prop prop_b	SW_Prop_A	?
502	Spot Weld	PARTPAT	Axis Nodes	Prop prop_a	Prop prop_b	SW_Prop_A	?
503	Spot Weld	PARTPAT	Axis Nodes	Prop prop_a	Prop prop_b	SW_Prop_A	?
504	Spot Weld	PARTPAT	Axis Nodes	Prop prop_a	Prop prop_b	SW_Prop_A	?

Connector ID: 501 Surface Patch A: Prop prop_a, Patch nodes: 381 382 402 403 423 424

Reset Cancel

Displaying Connectors

The display of connectors shall be that of a 3D marker (sphere) centered at the Connector location (midway between the two piece locations GA and GB), and a 2D marker (bar) connecting the GA and GB points. Connectors are posted to groups, like any other FEM entity, and their display shall be controlled using existing Patran tools.



This display shall be renderable in wireframe, hidden line, and solid shaded modes. The display attributes for Connectors (sphere color, size, etc.) are described **Display>Finite Elements** (p. 295) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

CHAPTER

13 The Modify Action

- Introduction to Modification
- Modify Forms

13.1 Introduction to Modification

The purpose of modification is to change one or more attributes of nodes, elements, and or MPCs which have been created, using one of the Create options in the Finite Element application.

Node modify options can affect the ID numbering, location, or the associated analysis and reference coordinate frames of an individual node or a group of nodes.

Element modify options can affect ID numbering, element topology (linear or higher order), or nodal connectivity (manual assignment or reversal of current connectivity).

Bar modify can split a bar element in two.

Tria modify can split a tria element into a pattern of two to four elements.

Quad modify can split a quad element into a pattern of two or four elements or NxM quad elements.

The MPC modify option can be used to add, modify, or delete terms of a currently existing MPC. Attributes of a term that can be modified include the sequence, coefficients, nodes, and the degrees-of-freedom.

Mesh modify, mesh smoothing is an iterative algorithm that can be used to optimize the shape of elements in an existing finite element mesh. Two principle uses for this feature are:

1. To more mesh nodes to the locations of “hard points” and then smooth the modified mesh. Hard points might be the locations of attachments or boundaries of holes.
2. Alter the default setting of a mesh smoothing parameter and then re-smooth the mesh. (Any transition mesh is smoothed automatically when originally created. In most cases, the default parameters yield an acceptable mesh.)

Mesh Seed modify allows the user to modify mesh control from one type to another without having to delete the old one and create a new one in place of the old one. This feature is particularly useful when user needs a node at a certain location when the edge has already been seeded with a certain type.

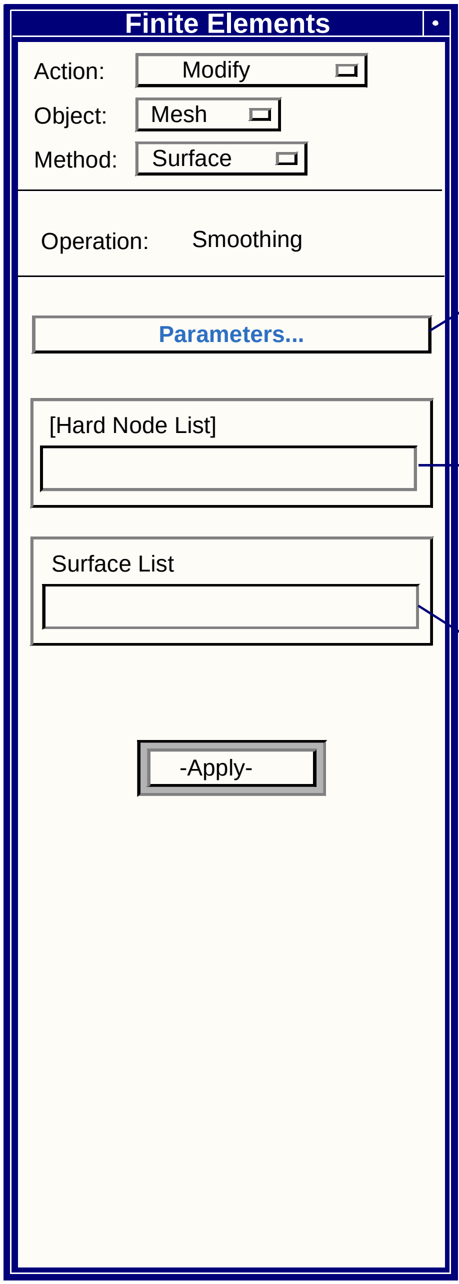
13.2 Modify Forms

When *Modify* is the selected action, the following options are available.

Object	Type	Description
Mesh	Surface	Improve an existing surface mesh with optional hard nodes.
	Solid	Improve an existing solid mesh with optional hard nodes.
	Sew	Stitches gaps on a mesh.
Mesh Seed	Mesh Seed	Allows modification of mesh seed on curves/edges.
Element	Edit	Changes attributes such as ID numbering, element topology, or nodal connectivity of selected elements.
	Reverse	Reverses the connectivity definition (and therefore the normal direction) of selected elements.
	Separate	Adds nodes to specified elements an separates them from the rest of the model.
	Shell Orientation	Orients elements in a model in the same direction.
Bar	Split	Splits a bar element in two.
Tria	Split	Splits a tria element into two to four elements.
Quad	Split	Splits a quad element into two to four elements.
Tet	Split	Splits a tet element.
Node	Move	Changes a nodal location.
	Offset	Moves nodes by an indicated vector distance.
	Edit	Changes attributes such as ID numbering, associated analysis and reference coordinate frames, or physical location of selected nodes.
	Project	Project nodes onto Surfaces, Curves or a constant coordinate plane (e.g X = 5).
MPC		Changes the attributes of a selected MPC.
Connector	Spot Weld	Changes the attributes of a Spot Weld Connector.

Modifying Mesh

The smoothing algorithm used is the iterative Laplacian-Isoparametric scheme developed by L. R. Herrmann. The final mesh and the execution time for smoothing are controlled by the Smoothing Parameters.



Brings up the *Smoothing Parameters* form to change smoothing parameter values. These are the same parameters that are used during mesh creation.

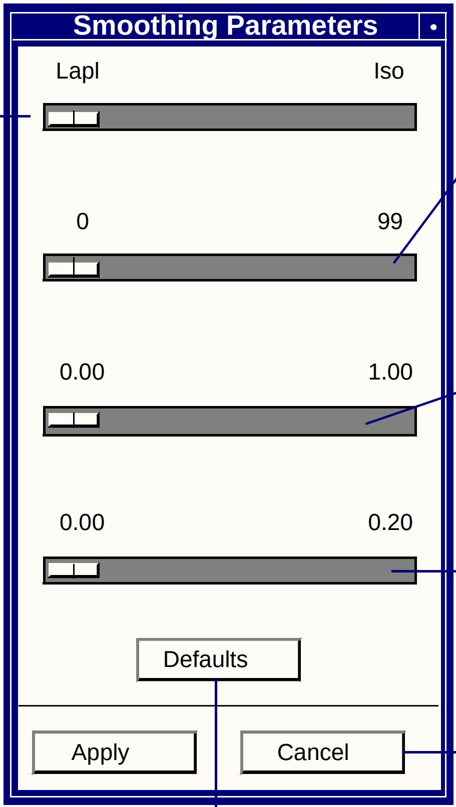
Specifies nodes which will remain stationary during smoothing. Nodes around the perimeter of each surface will be treated as hard nodes automatically. This function is optional.

Selects surface(s) to be smoothed. Each surface will be smoothed individually. All elements associated with a surface will be included in the smoothing operation. Elements may have been created by IsoMesh or Paver, or they may have been created or modified by element editing.

Smoothing Parameters

Figure 13-1

Sets weighting factor for combining Laplacian and Isoparametric smoothing. At the extreme left, pure Laplacian smoothing is used; at the extreme right, pure Isoparametric smoothing is used; at intermediate positions, a combination of the two methods is used. Pure Laplacian smoothing was selected as the default Smoothing Factor because it produces the best mesh in most cases. One notable exception occurs when a surface has significant inplane curvature. In this case, Laplacian smoothing pulls the elements toward edges with inner curvature. Isoparametric smoothing is usually better in this case. See **Figure 13-2**.



Sets the maximum number of iterations allowed for mesh smoothing. Default value is 20.

Sets the smoothing acceleration factor. This factor increases the distance each node is moved during a smoothing cycle and may thus allow smoothing to converge in fewer cycles. For example, a value of 0.5 causes each node to move 50% farther than computed at each iteration. However, caution is advised because this factor can also make the smoothing process diverge, particularly if the geometry is highly curved or skewed. Therefore, a warning is given whenever this factor is set above a value of 0.1.

Controls the smoothing termination tolerance factor. Default value is 0.05, where smoothing stops when the largest distance moved by any node during the previous iteration is less than 5% of the shortest element length along an edge of the geometry.

If selected, MSC.Patran will reset the smoothing parameter values back to the original set of values that existed upon entry to the Mesh Parameters form.

If selected, the smoothing parameters will be reset back to the original *factory* default values. These are: Smoothing Factor = 0.0, Maximum Cycles = 20, Acceleration Factor = 0.00, Termination Factor = 0.05.

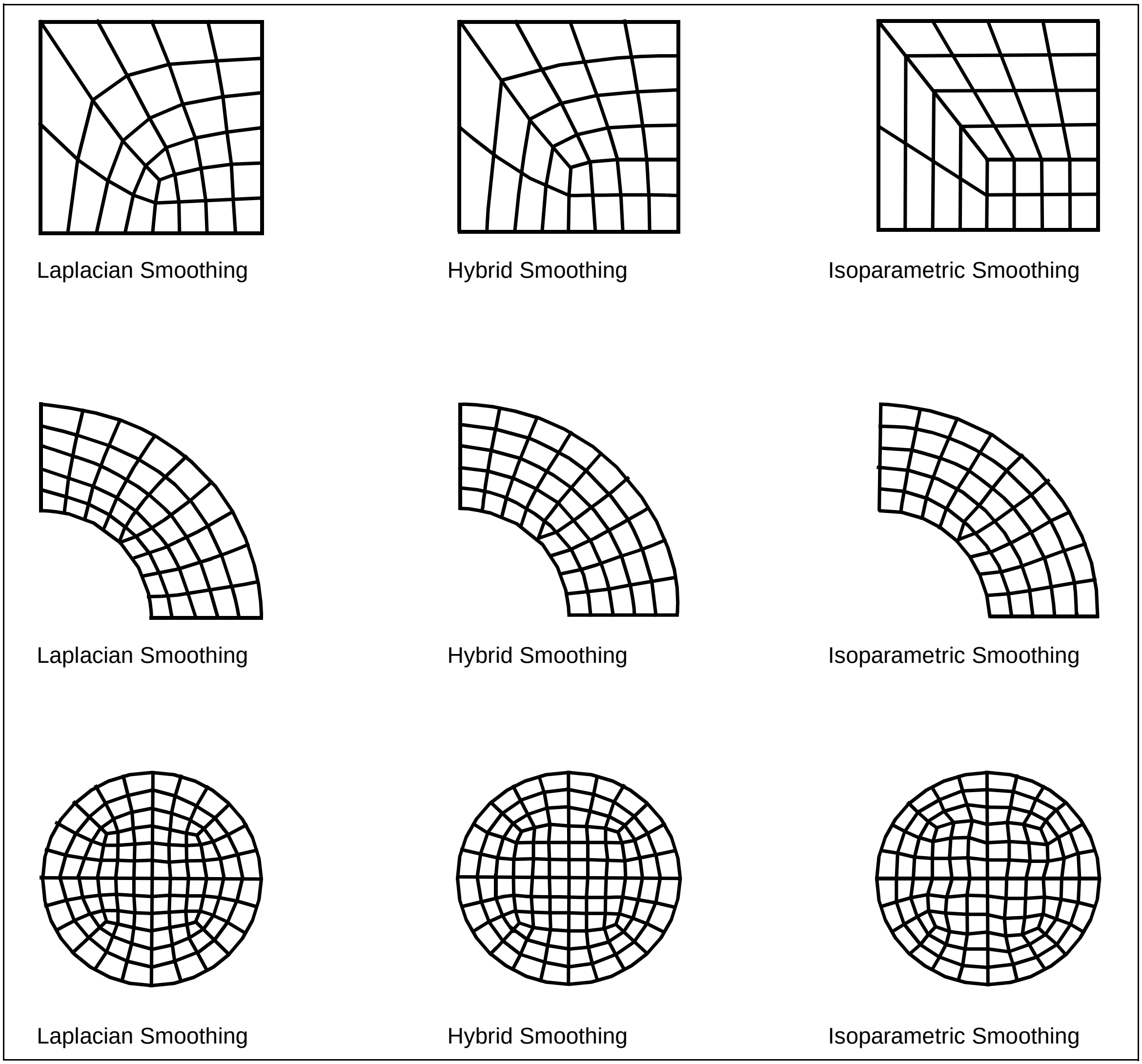
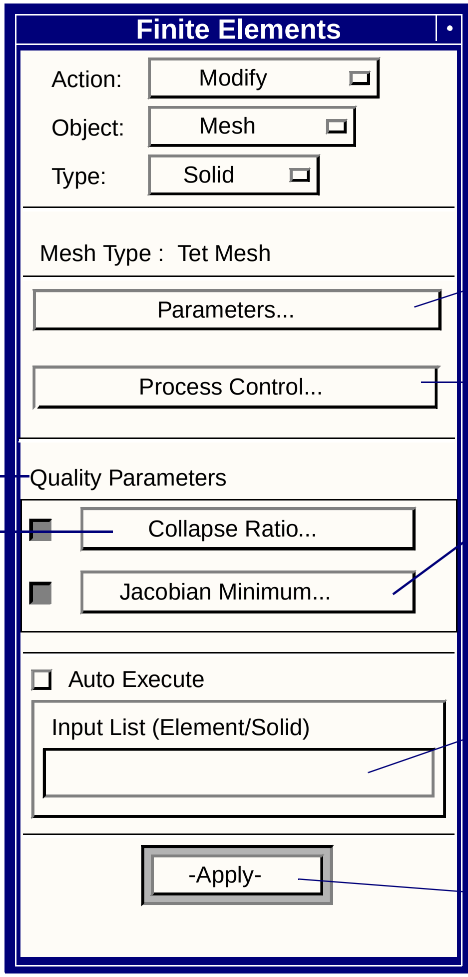


Figure 13-2 Example Meshes with Different Values of Smoothing Factor

Mesh Improvement Form

The purpose of this form is to improve the quality of a solid mesh with respect to the criterion selected.



Sub-form for specific parameters. The user would typically not need to modify these parameters. See **General Parameters** (p. 272) for more information.

Sub-form for controlling the process of improving the mesh. The user would typically not need to modify these parameters. See **Process Control** (p. 273) for more information.

Sub-form for parameters related to the minimum Jacobian value inside an element. The user would typically not need to modify these parameters. Setting the toggle as enabled will cause the element quality to consider this criterion. See **Jacobian Minimum** (p. 275) for more information.

List of elements or solid. For supplied solids, the associated elements will be considered. Duplicate entities are automatically removed.

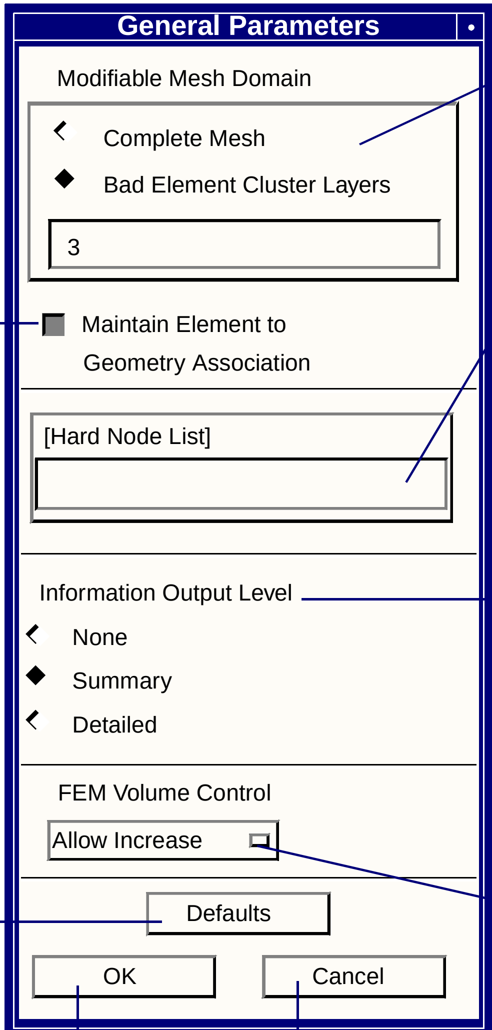
Submit the functionality.

Sub-form for parameters related to the tetrahedron collapse ratio. The user would typically not need to modify these parameters. Setting the toggle as enabled will cause the element quality to consider this criterion. See **Collapse Ratio** (p. 274) for more information.

List of criterions that define the quality for the elements to be considered. The quality of each element is determined by the worst normalized value for all considered criterions in this frame.

General Parameters

Enabled in order to consider geometric boundaries and create new node/element geometric association. When disabled, the solid mesh will still consider its boundary, but ignore delimitation of the surfaces which define the solids to which the elements may be associated. The later (disabled) is the same as if the envelope of the solid mesh corresponds to a single geometric surface.



Allows you to specify how many layers around each bad element you want to consider as a modifiable region. Or you can pick all of the mesh.

List of hard nodes, not to be modified.

Level of information on the bad elements initially in the mesh, during iterations, and those left in the end (if any):

None: Only the percentage of completion forms are displayed while the process is running.

Summary: In addition to information displayed under "None", the history window and session file will indicate the elements failing the criterion in the original state of the mesh, a status line for each iteration, a table of failing elements in the final state of the mesh and some advice if applicable.

Detailed: In addition to information displayed under "Summary", exact references are given for the elements reference in each iteration status line.

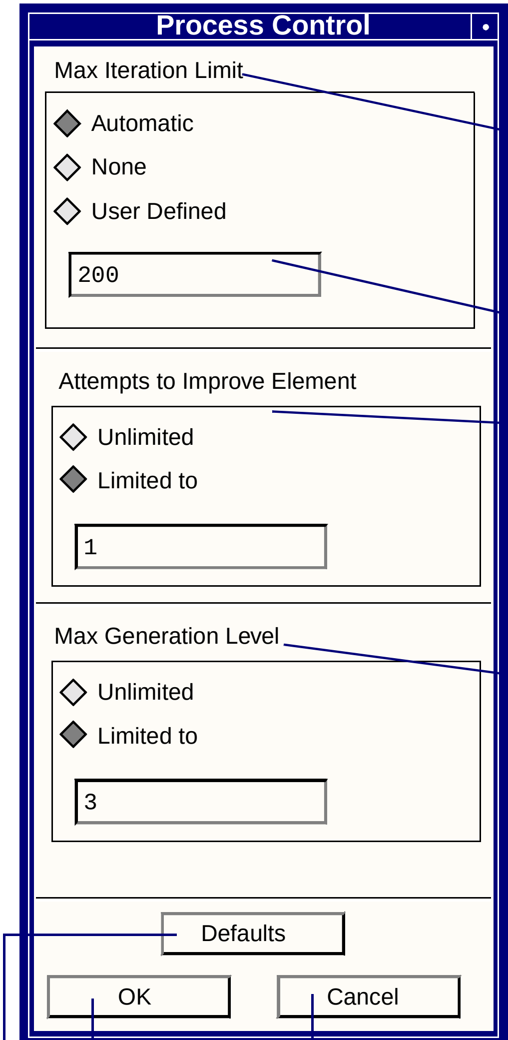
FEM Volume Control for non-snapping element edges on mesh boundary: This only applies for edges on the mesh boundary without an associated geometry of lower order than that of the mesh. Choices are : "Allow Increase", "Allow Decrease", "Constant", "Any Change".

Exit this form without considering changes.

Accept the changes and close this form.

Reset default values.

Process Control



Maximum number of iteration to perform. The process can terminate before in the advent of the mesh to satisfy all quality criterions.

Automatic: The limit is automatically determined based on the quantity of bad elements:

=100 x (number of bad elements)
200 minimum, 1000000 maximum

None: No practical limit (set to 10 million).

User Defined: The limit is specified in the databox below. A value of 0 will cause the mesh to be evaluated but no modification will be performed. A value of N (0<N) will allow of a maximum of N mesh modification attempts.

Attempts to Improve Element. The number of times an element will be attempted to be improved (replaced by better ones). The algorithm allows the improvment process to return to a element which was previously unfixable (could not be replaced by better ones).

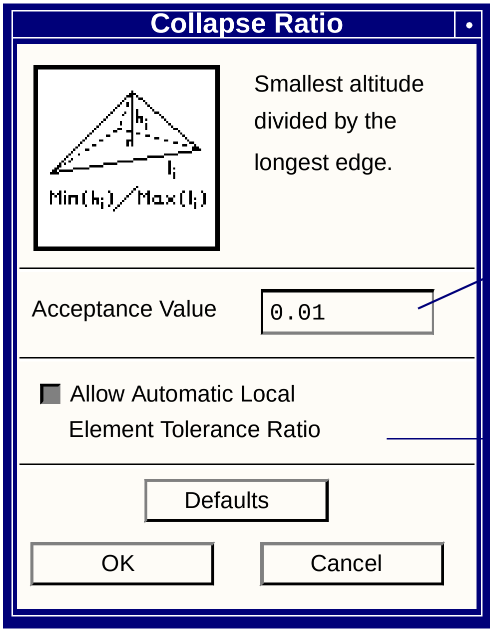
Thissets the limit for the number of times that an element can by replaced. The first time an element is replaced by a set better elements, those new elements are of generation 1(the original were generation 0). Further replacement of generation 1 elements will lead to generation 2 elements and so on.

Exit this form without considering changes.

Accept the changes and close this form.

Reset default values.

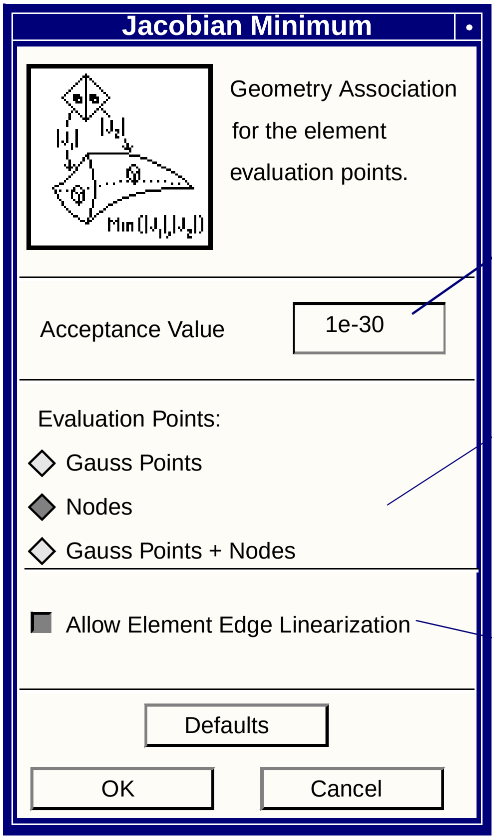
Collapse Ratio



The acceptance value for the selected criterion. This is typically 0.01 for a subsequent NASTRAN analysis. Normalized value will be established from this value and ultimate limits : 0.0 and sqrt(2/3).

When enabled, allows to find another value for the "Local Element Tolearnce Ratio" for individual element which can not be improved with the value supplied. It is recommended to always enable this option.

Jacobian Minimum



The acceptance value for the selected criterion. This is normally set to a small value above zero to ensure positive Jacobians inside the elements. Normalized value will be established (using am inverse tangent function) from this value and ultimate limits : +/-infinity.

Points inside the element where the Jacobian will be evaluated :

Gauss Points : Only the Gauss-Legendre quadrature integration points are considered.

Nodes : Only the element's nodes are considered.

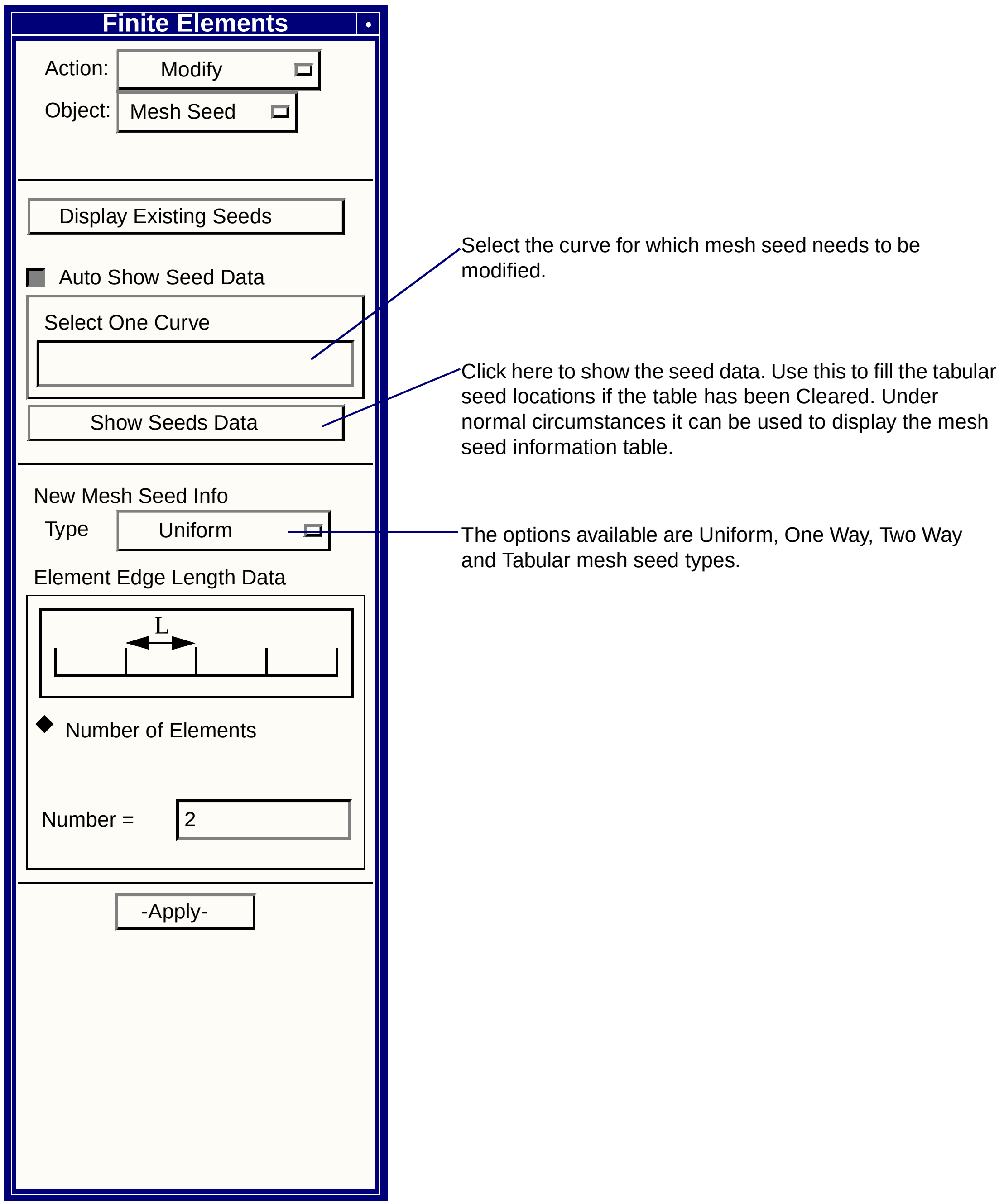
Gauss Points + Nodes : Both previous location sets are considered. This corresponds to the Verify/Element/JacobianZero form.

The "Allow element edge linearization" feature enables the improvement process to linearize element edges as a last resort to fix otherwise unfixable (within specified process control limitaions) elements because of an unacceptable "Scaled Jacobian Minimum" quality criterion.

The linearization process in the "Mesh Improver" linearizes the most curved edge of a bad element, then re-checks the quality before proceding to other edges. By this method we minimize the amount of linearized edges.

Modifying Mesh Seed

The modify mesh seed menu allows users to change mesh seed types.



Sew Form

Using Modify / Mesh / Sew form sews gaps on a mesh consisting of all tria3 elements. This program removes interior free edges on a mesh by merging nodes and splitting triangles automatically (see Figure 13-3 and Figure 13-4).

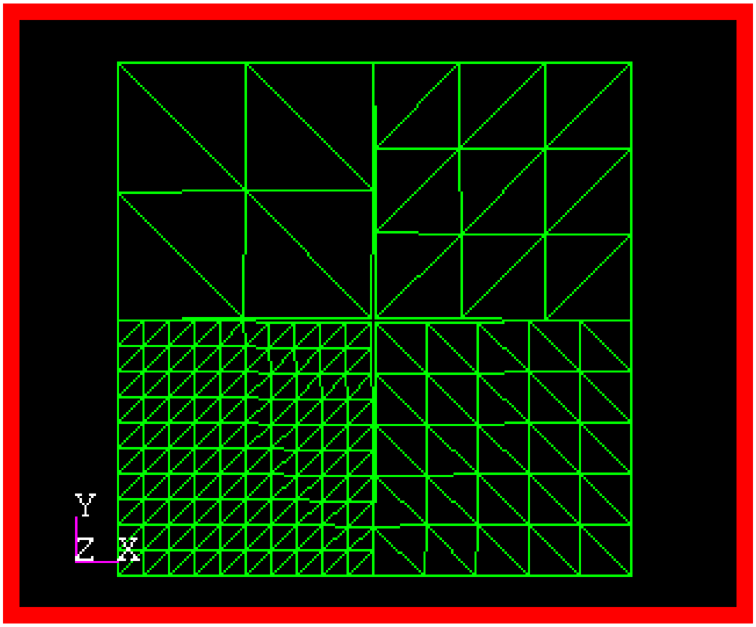


Figure 13-3 Mesh Before Sewing

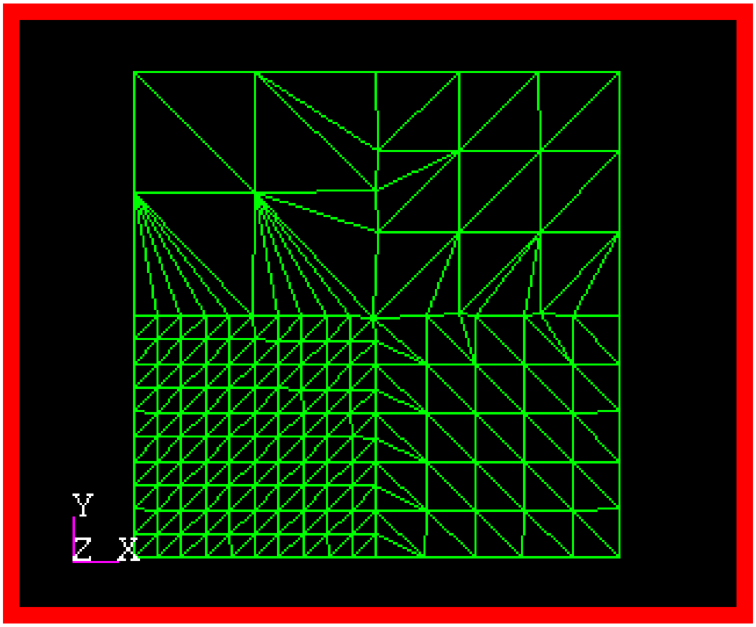
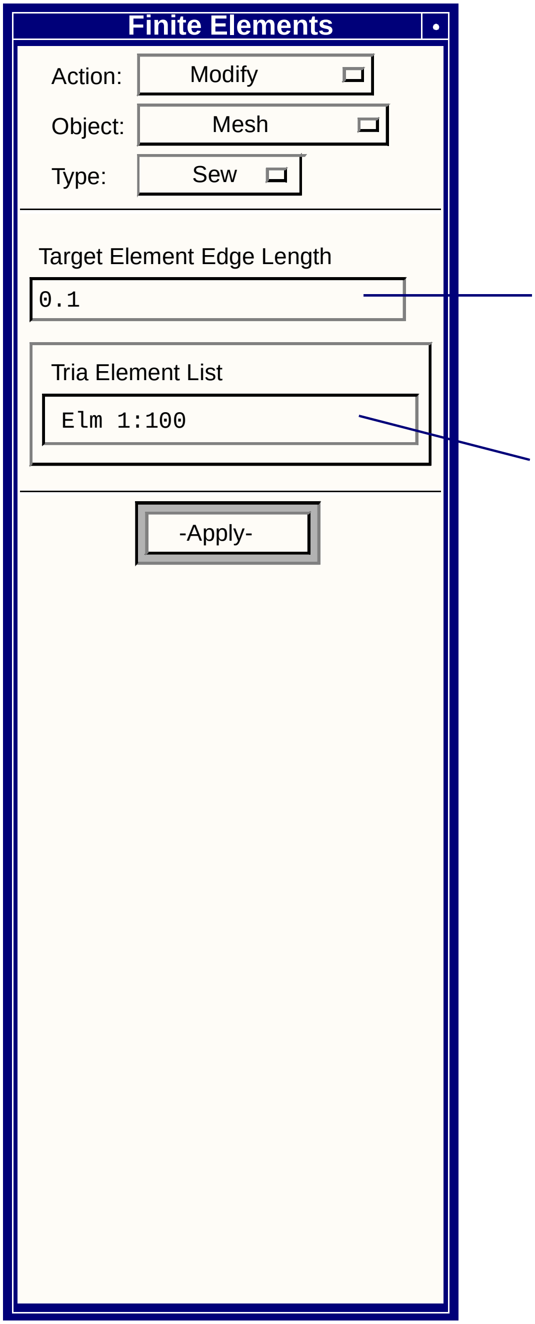


Figure 13-4 Mesh After Sewing

The primary purpose of this program is to provide users a useful tool to obtain a congruent mesh which will be used to create a tessellated surface. (See **Created Tessellated Surface from Geometry Form** (p. 306) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*.) For this reason, the elements modified or created by this program may not have very good quality.

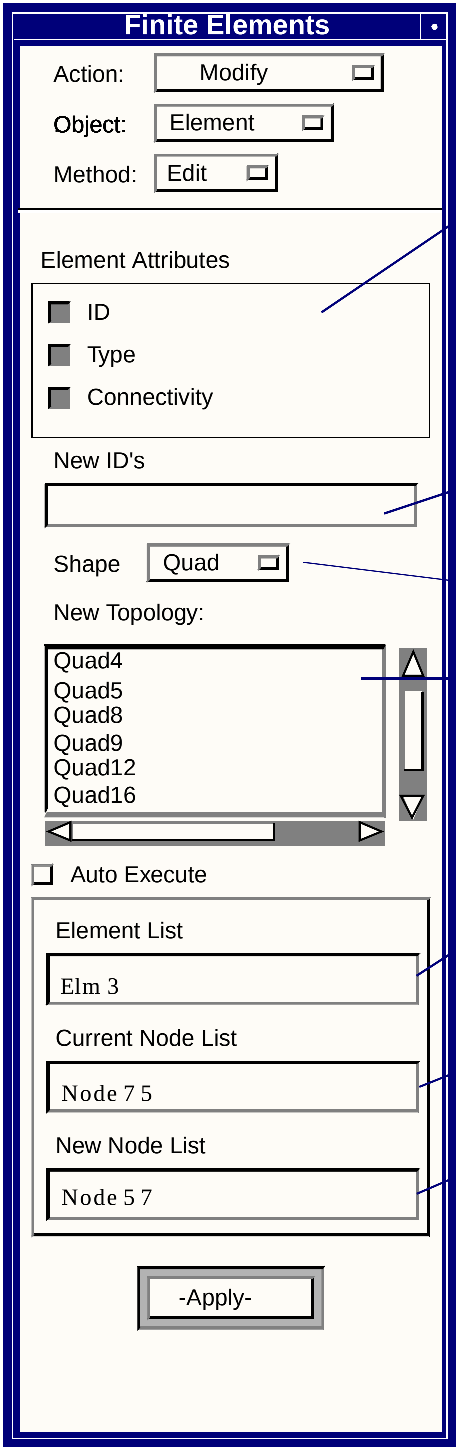


Specify the target element edge length which was used as a global edge length when creating the mesh. The gap tolerance is equal to one tenth of the target element length.

Specify a set of tria elements consisting of the mesh to be sewed.

Modifying Elements

Edit Method



When one of these attributes is selected, the appropriate information to modify appears on the form. When this form is initially opened, none of the attributes are selected; therefore, only the Element Attributes toggle is initially displayed. Any combination of these attributes may be selected. If only Connectivity is selected, a toggle will become visible to modify all elements associated to the current nodelist.

For information regarding this toggle see **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Specifies shape of the new elements.

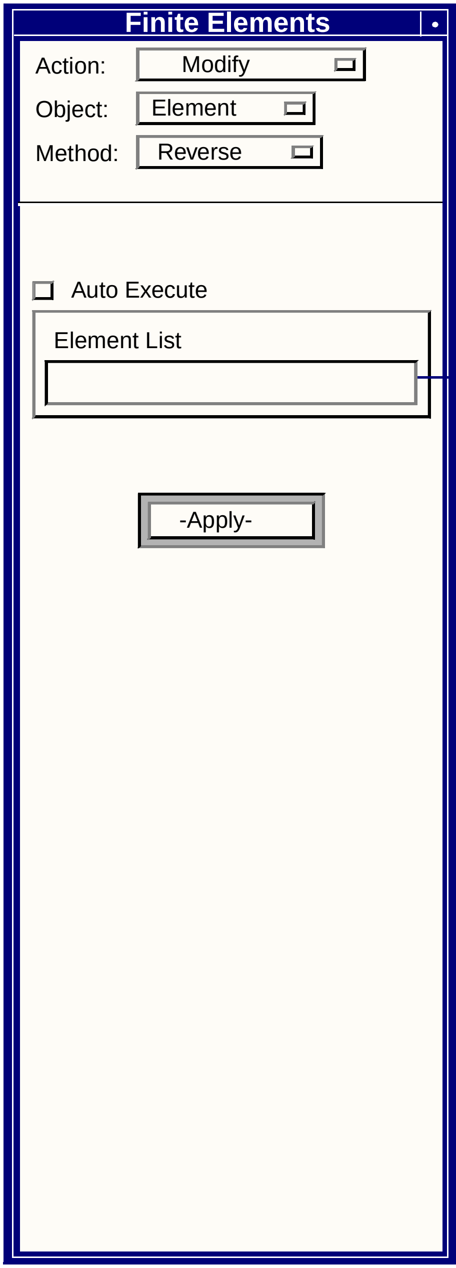
Specifies topology of the new elements.

This box appears on the form when any element attribute is selected. It lists the ID numbers of the elements to which the changes apply.

Lists the nodes to be modified for each element.

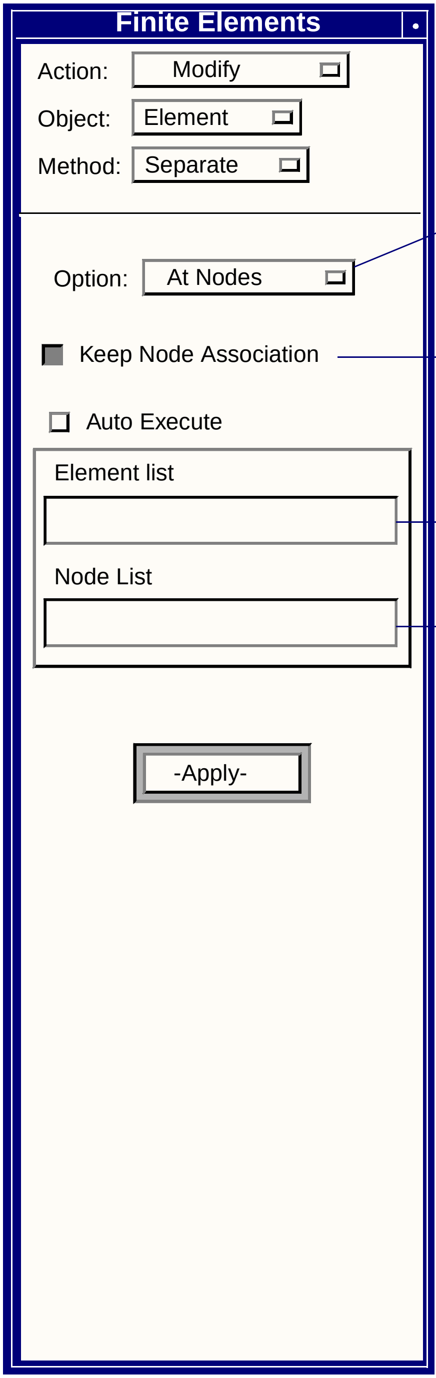
The list of nodes which are to replace those in the Current Node List. There is a one-to-one correspondence between the Current Node List and the New Node List. In the example shown, Nodes 5 and 7 are to be swapped in the connectivity order for Element 3.

Reverse Method



Specifies the list of elements whose connectivity, and therefore normal direction, is to be reversed.

Separate Method



The following options are available:

At Nodes: separate elements at nodes

At Elem Edges: separate elements at edges

At Elem Faces: separate elements at faces

At Free Edges: separate elements at free edges

At Free Faces: separate elements at free faces

Toggle to specify if the new nodes will keep the same association to geometry as the old ones.

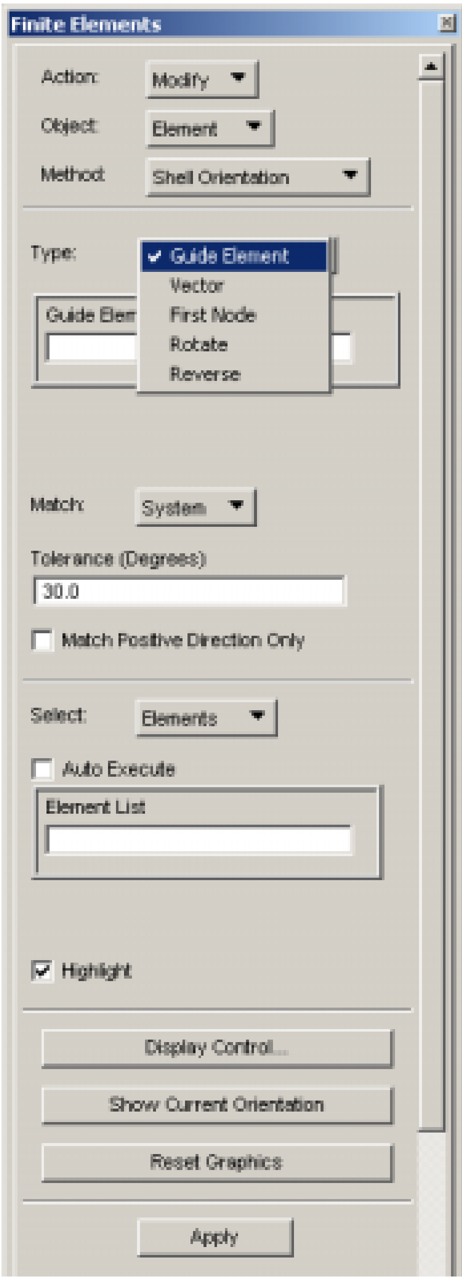
Specifies the list of elements to be separated.

Specifies the list of nodes. For each node a new node will be created and the element connectivity will be updated for the specified elements. The Node list box will only be visible for option At Nodes. For all other options the nodes associated to the specified entities will be used instead.

Shell Orientation

A shell element's orientation and normal are based solely on its connectivity, shape and spatial orientation. There is no element property that affects a shell element's orientation or normal. Therefore, the only model changes made using this functionality will be shell element connectivity.

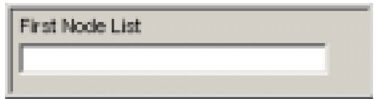
Since a shell element's connectivity is the only model change that can be made to modify its orientation and normal, the degree of control is limited. Depending on its shape and spatial orientation, there may not be a way to obtain the exact orientation. In these cases, the closest match will be made.



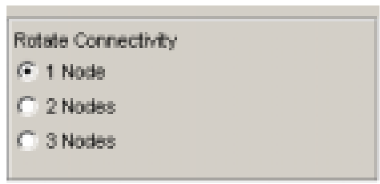
Type

Type consists of five options for specifying how to reorient the selected elements.

- **Guide Element** — Uses the orientation of an element.
- **Vector** — Uses any valid vector specification. This includes any axis of a selected coordinate frame.
- **First Node** — Changes the element's connectivity such that the first node is the one specified. A list of first nodes is provided.



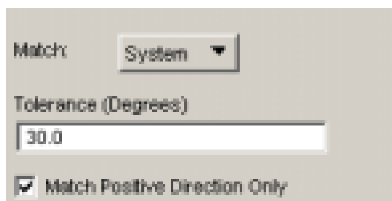
- **Reverse** — The normals of the selected elements will be reversed.
- **Rotate** — Rotates the element's connectivity by 1, 2 or 3 nodes.



Guide Element

For Guide Element and Vector, the coordinate system or normal may be matched.

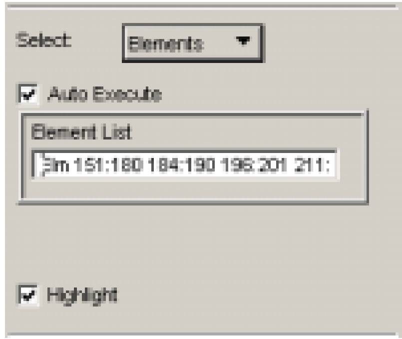
- **Match System** — For Match System a tolerance angle may be specified. If the tolerance angle is exceeded, for any element, that element's orientation is not modified.



Select

Select consists of three options for selecting the elements to be reoriented.

- Elements — The Elements option allows individual shell elements to be selected. Auto Execute may be used to have processing occur immediately after each update of the element list.



- Groups — The Groups option allows any number of groups to be selected. The shell elements in the groups will be processed. The Highlight toggle can be used to turn on/off highlighting of the selected elements.

- Current Group — The Current Group option allows the current group to be selected. The shell elements in the current group will be processed. The Highlight toggle can be used to turn on/off highlighting of the selected elements.

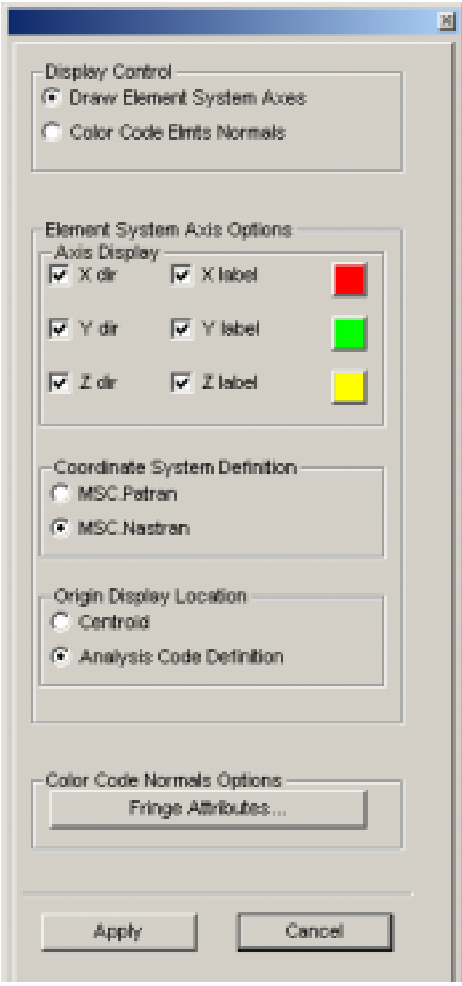
Display Control — The Display control will control the way that elements are displayed. See Display Control (p. 285) for more information.

Show Current Orientation — Displays element orientation and normal as specified in the Display Control form on the selected elements.

Reset Graphics — Removes element orientation and normal graphics.

Apply — Applies the specified changes and updates the element orientation and normal display.

Display Control



Display Control
Element system axes (Z = normal) can be drawn as vectors and element normals can be color coded in a fringe plot. Either or both displays can be selected.

Element System Axis Options
Any combination of axes, labels and colors may be selected.

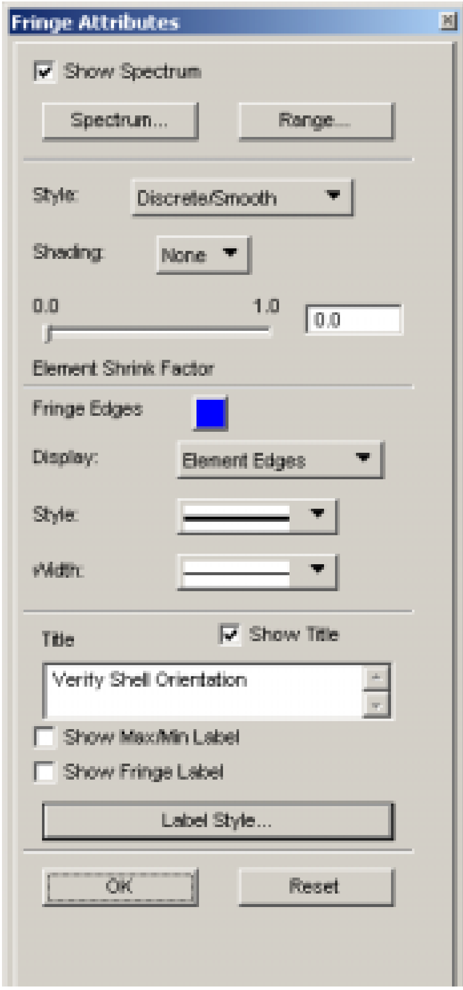
Coordinate System Definition
Either MSC.Patran or MSC.Nastran conventions can be chosen for displaying the element system. No other analysis code conventions are currently available. Beam axes can include or ignore offsets.

Origin Display Location
The origin of the element system axes can be placed at the element centroid, or the analysis code's (MSC.Patran / MSC.Nastran) definition.

Color Code Normals Option
Brings up the Fringe Attributes form for controlling the color coded fringe plot attributes. See Fringe Attributes (p. 286).

Fringe Attributes

This form controls the attributes of the color coded element normal fringe plot.



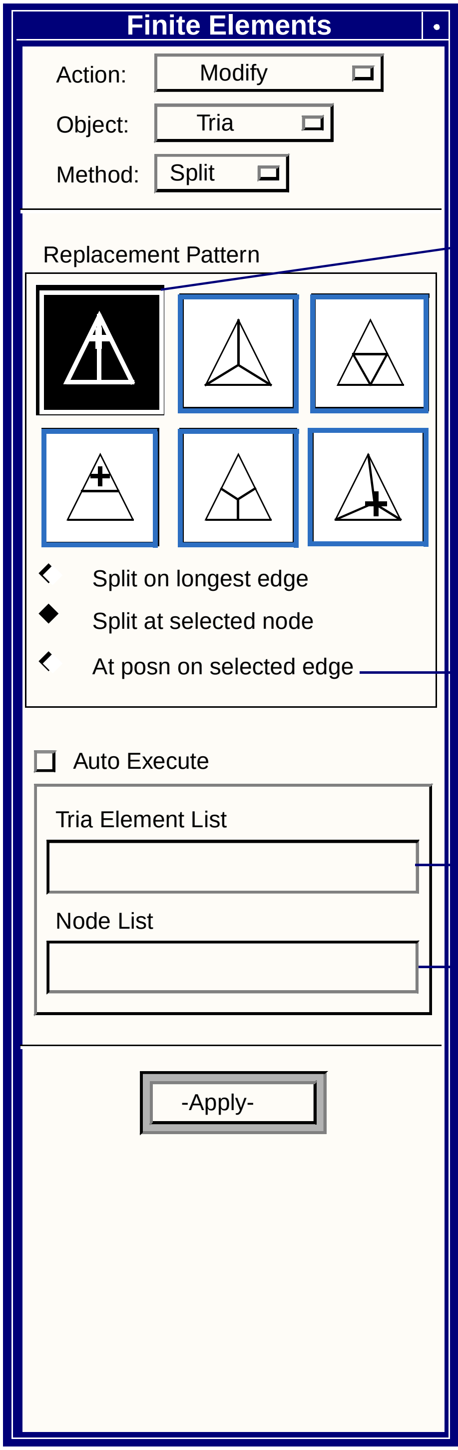
Modifying Bars

Specifies the list of bar elements to split.

Note: The new bars will have the same topology as the parent (i.e., a Bar3 will be split into two Bar3s).

Modifying Trias

Splitting a Tria into Two Trias



Select this icon to enable the two tria option.

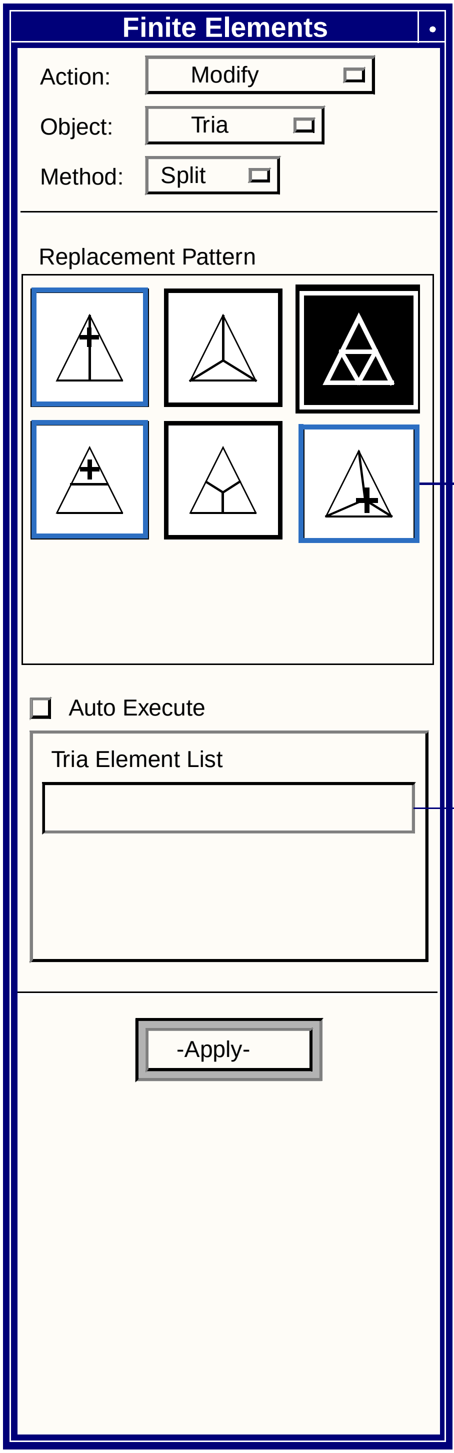
Selects a position on an edge.

Specify the trias to be split by selecting from the graphics window or entering a list of elements.

Specify the node where the split is to occur. This list may contain one entry for each element or a single entry. If a single entry, each element will be split at the same corner relative to the element origin as the first element.

Note: The new trias will have the same topology as the parent (i.e., a Tria6 will be split into two Tria6s).

Splitting a Tria into Three Trias, Four Trias, or Three Quads

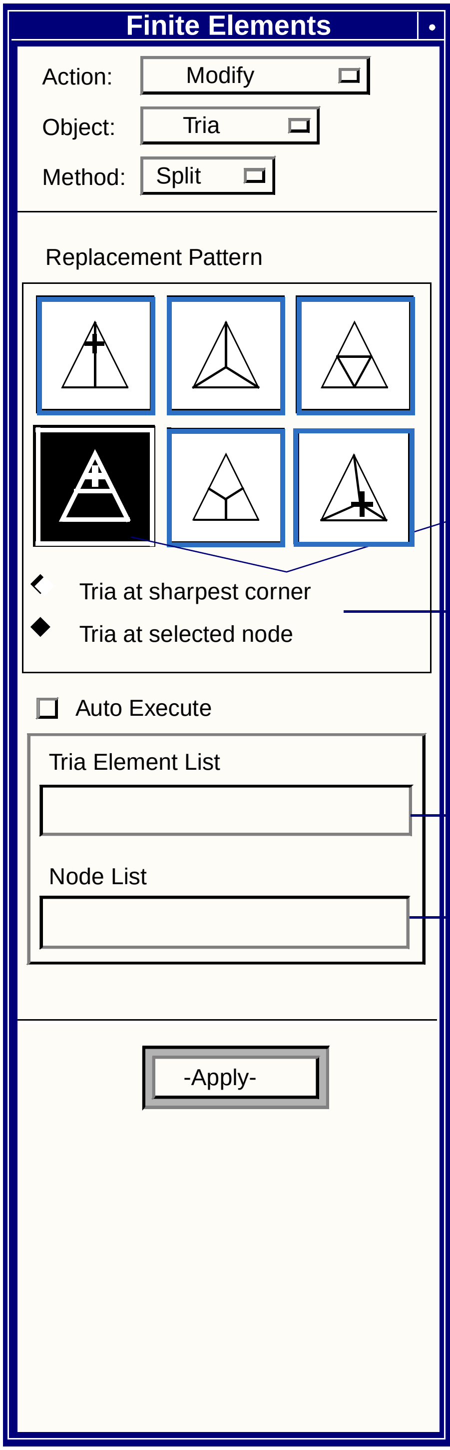


Select one of these icons to enable the selected pattern.

Specify the trias to be split by selecting from the graphics window or entering a list of elements.

Note: The new elements will have the same topology as the parent (i.e., a Tria6 will be split into Tria6s or Quad8s).

Splitting a Tria into a Tria and a Quad



Select this icon to enable the tria-quad option.

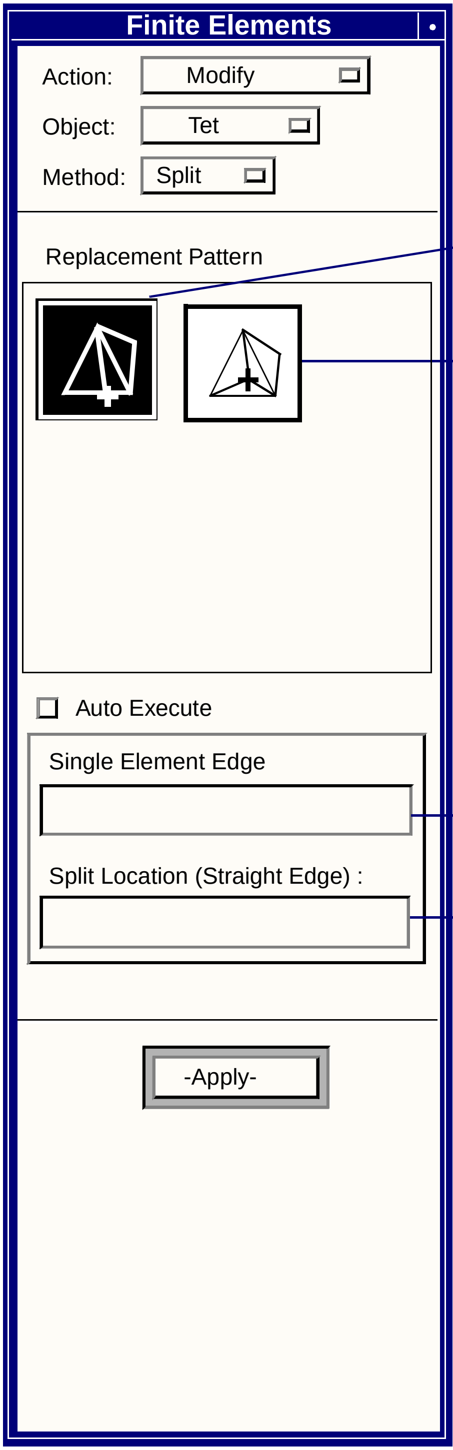
Select automatic or manual operation.

Specify the trias to be split by selecting from the graphics window or entering a list of elements.

Specify the node to orient the split. This list may contain one entry for each element or a single entry. If a single entry, each element split will be oriented the same relative to the element origin as the first element.

Note: The new elements will have the same topology as the parent (i.e., a Tria6 will be split into a Tria6 and a Quad8).

Splitting Tet Elements



Split a tet element at a selected edge.

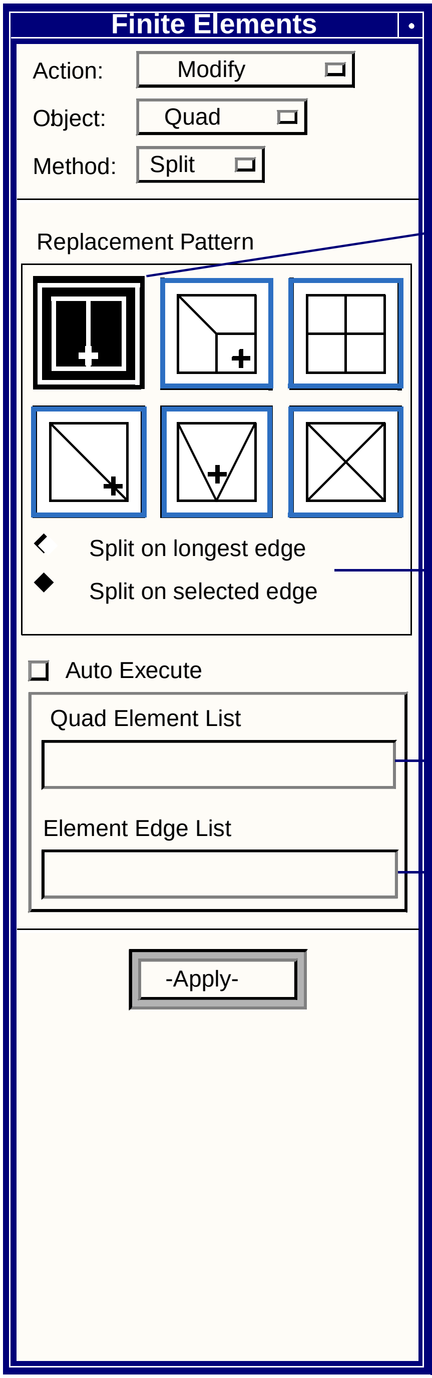
Split a tet element at a selected face location.

Select the tet edge to split.

Select the edge split location. This list may contain one entry for each element or a single entry. If a single entry, each element split will be oriented the same relative to the element origin as the first element.

Modifying Quads

Splitting a Quad into Two Quads



Select this icon to enable the two quad option.

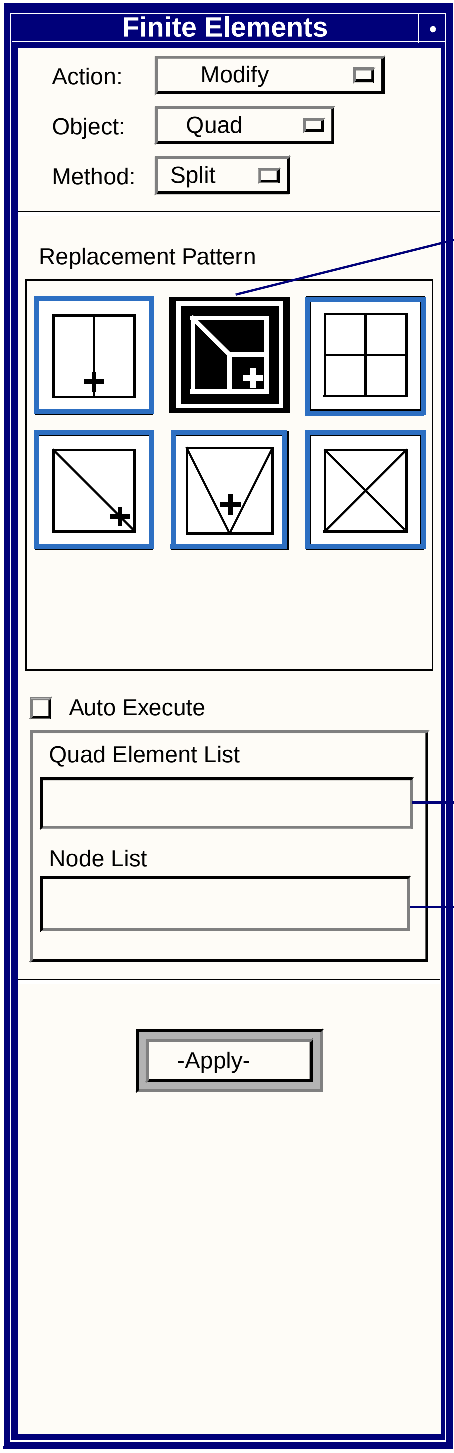
Select automatic or manual operation.

Specify the quads to be split by selecting from the graphics window or entering a list of elements.

Specify the edge where the split is to occur. This list may contain one entry for each element or a single entry. If a single entry, each element will be split at the same edge relative to the element origin as the first element.

Note: The new quads will have the same topology as the parent (i.e., a Quad8 will be split into two Quad8s).

Splitting a Quad into Three Quads



Select this icon to enable the three quad option.

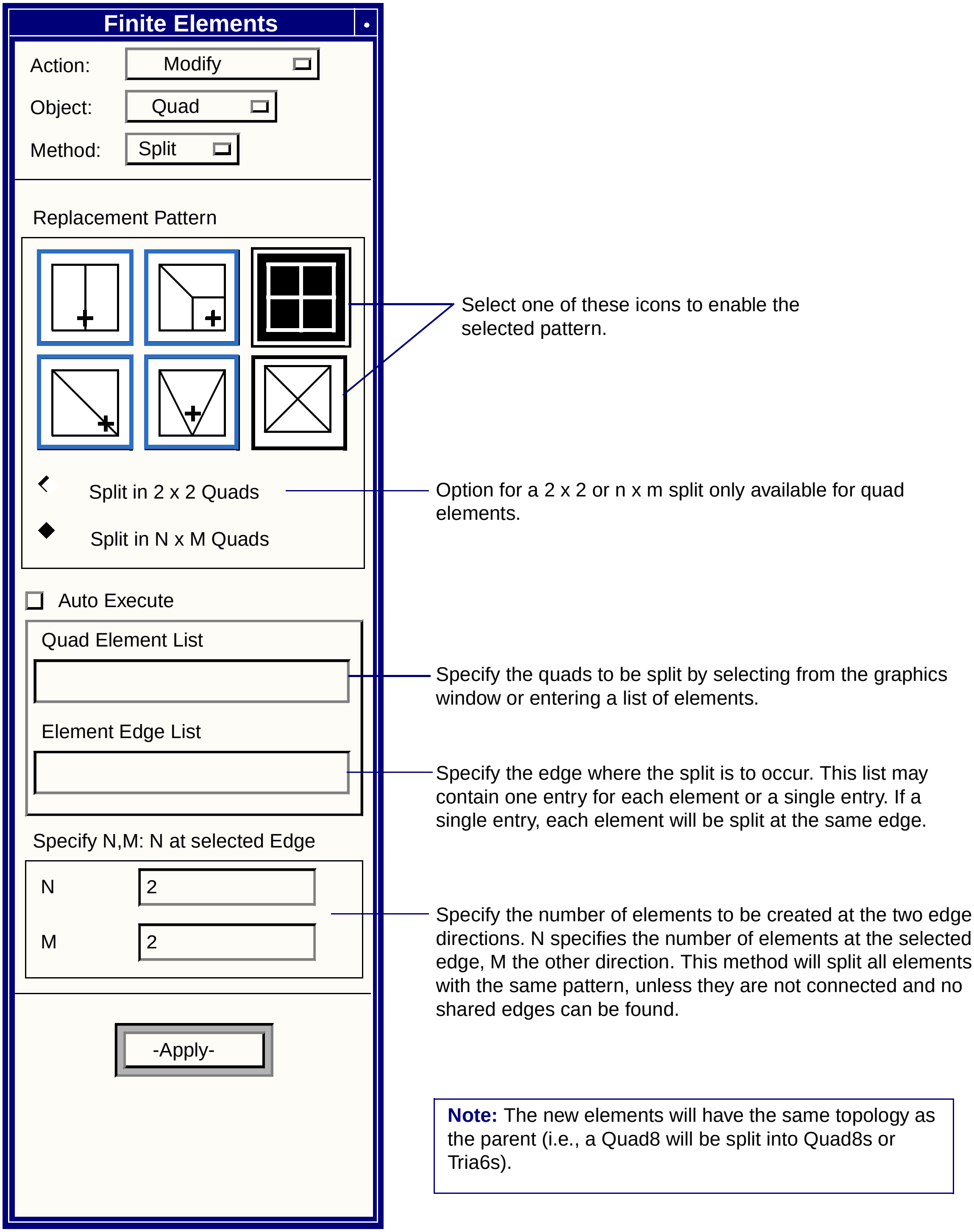
Specify the quads to be split by selecting from the graphics window or entering a list of elements.

Specify the node to orient the split. This list may contain one entry for each element or a single entry. If a single entry, each element split will be oriented the same relative to the element origin as the first element.

Note: The new quads will have the same topology as the parent (i.e., a Quad8 will be split into three Quad8s).

Splitting a Quad into Four Quads or Four Trias or NxM Quads

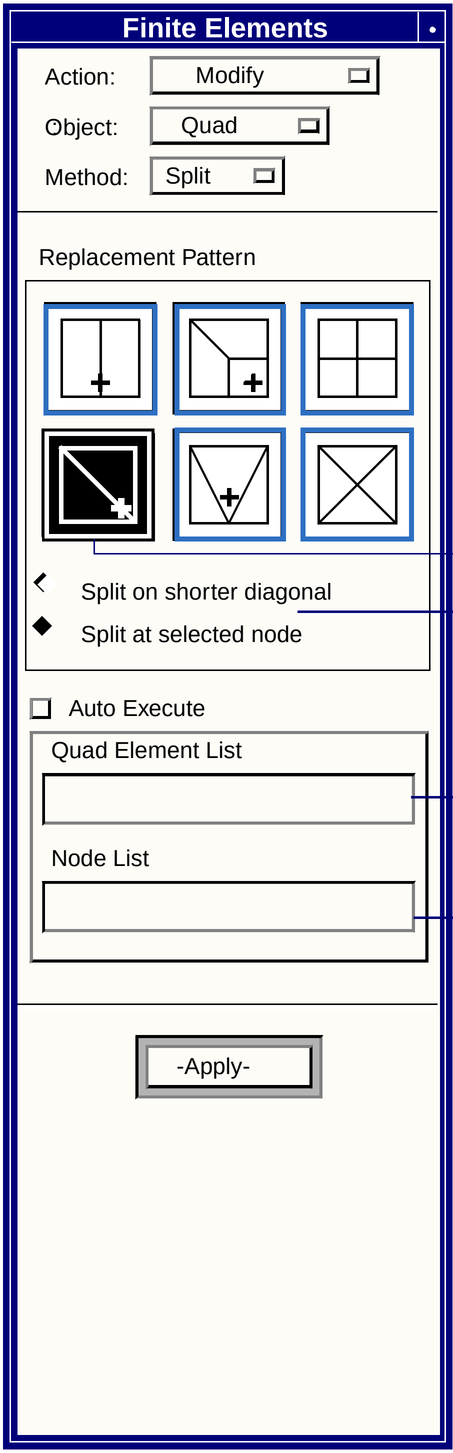
Figure 13-5



Note: The new elements will have the same topology as the parent (i.e., a Quad8 will be split into Quad8s or Tria6s).

Splitting a Quad into Two Trias

Figure 13-6



Select this icon to enable the two tria option.

Select automatic or manual operation.

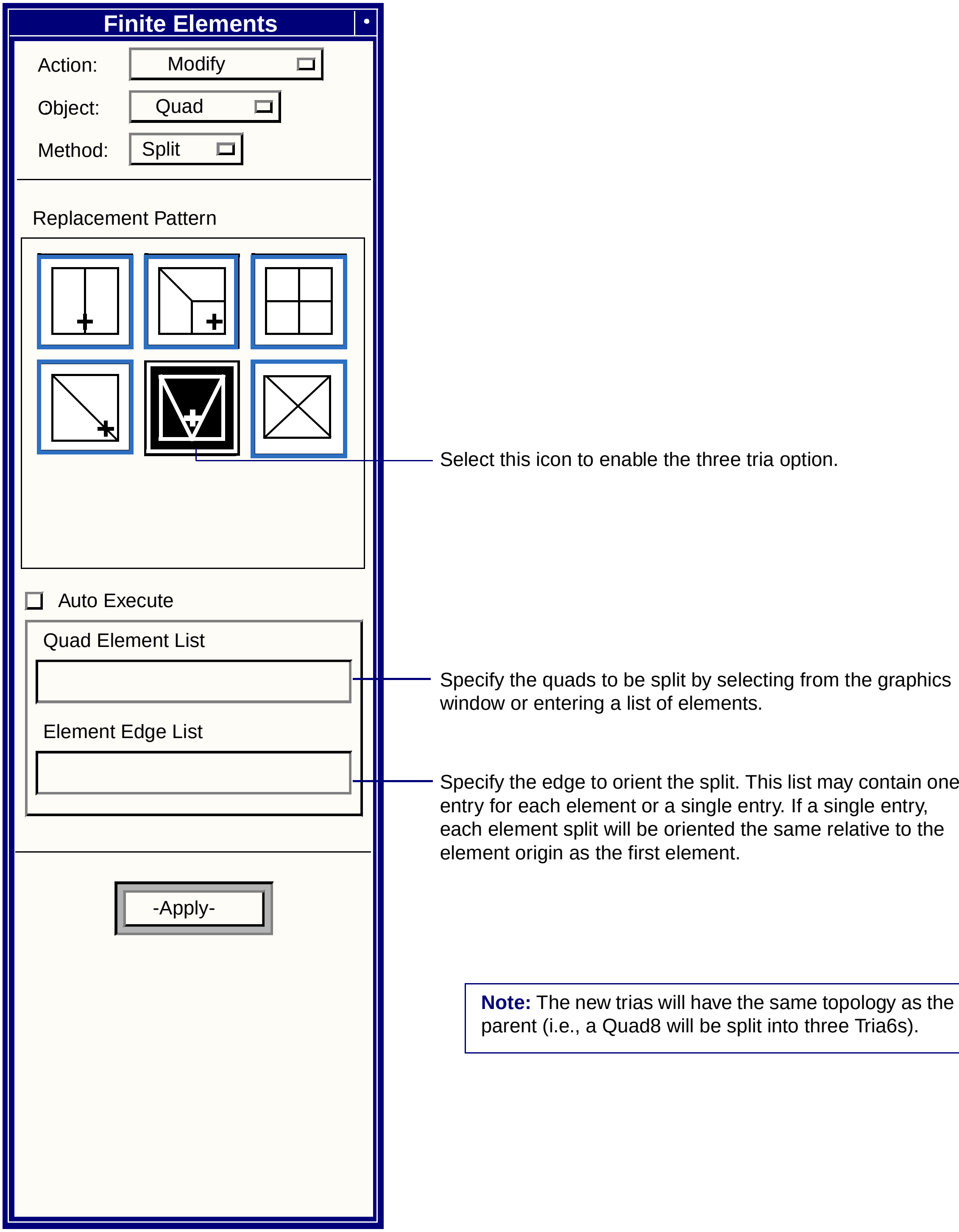
Specify the quads to be split by selecting from the graphics window or entering a list of elements.

Specify the node where the split is to occur. This list may contain one entry for each element or a single entry. If a single entry, each element will be split at the same corner relative to the element origin as the first element.

Note: The new trias will have the same topology as the parent (i.e., a Quad8 will be split into two Tria6s).

Splitting a Quad into Three Trias

Figure 13-7

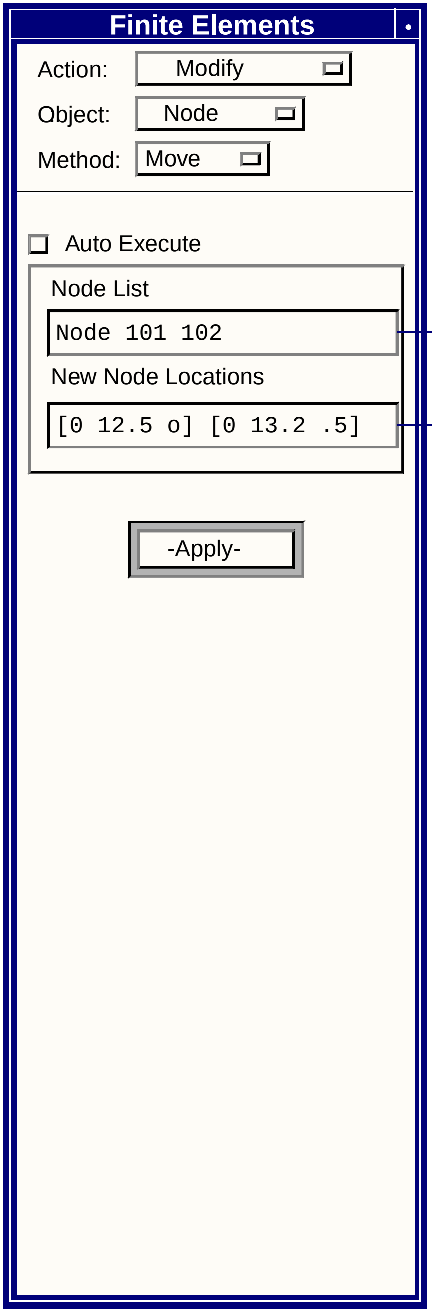


Note: The new trias will have the same topology as the parent (i.e., a Quad8 will be split into three Tria6s).

Modifying Nodes

Move Method

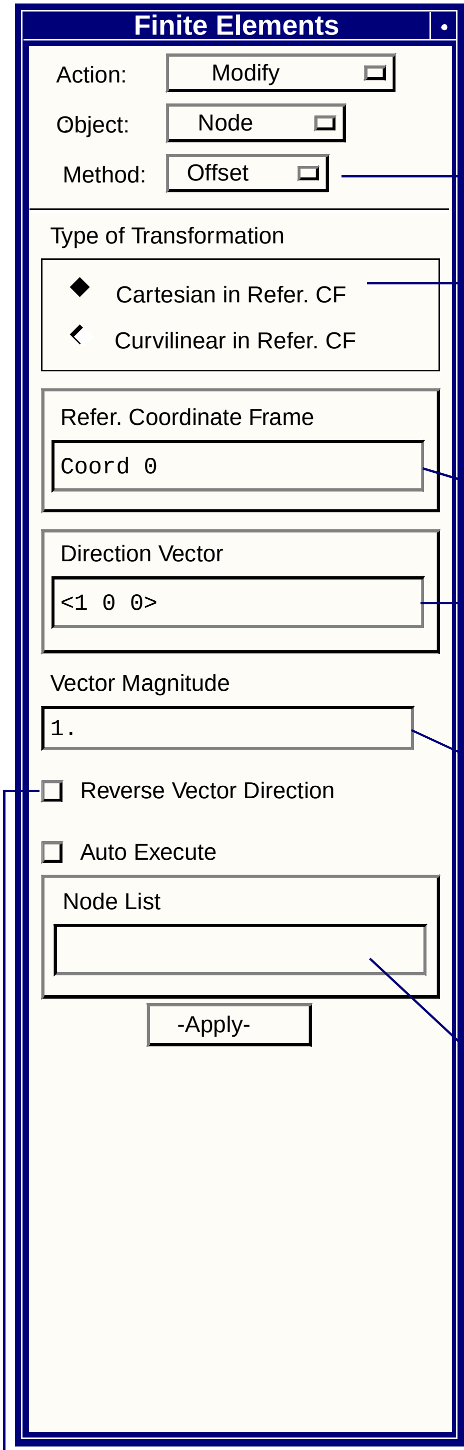
Figure 13-8



Specifies the list of nodes to be moved.

Specifies a list of new node locations by entering coordinates in the global rectangular system (as shown) or by using the select menu.

Offset Method



The Node/Offset method allows moving nodes by an indicated vector distance.

Specifies whether the offset direction is defined relative to rectangular coordinates in any selected coordinate frame, or relative to curvilinear coordinates of a selected cylindrical or spherical reference coordinate frame. If rectangular coordinates are selected, the vector may be selected via the select mechanism.

Specifies a *reference coordinate* frame in which the direction vector is defined.

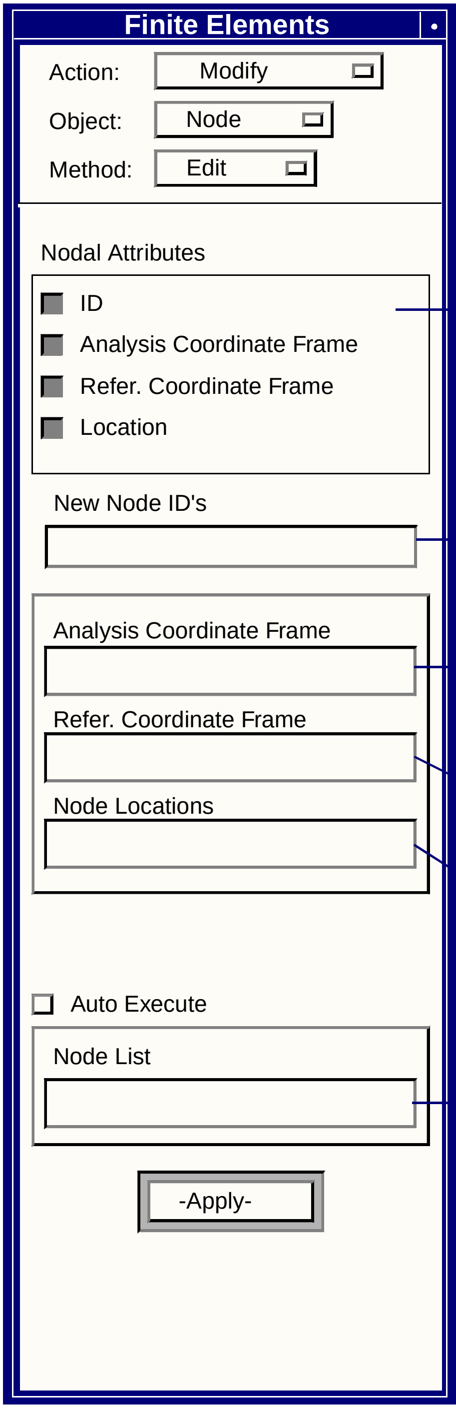
Specifies the direction in which the nodes are to be offset. For cartesian enter direction vector components along the three axes. For curvilinear enter actual incremental distances. For example enter Δr, $\Delta\theta$ (in degrees) and Δz.

For cartesian only, this specifies the actual distance the nodes are to be offset in the direction indicated by the direction vector. When the direction vector or the reference coordinate frame are modified, the magnitude of the new vector is automatically loaded here. The magnitude may then be modified, if a different length is desired. PCL expressions can be entered here to achieve convenient scaling of the original vector length (i.e. ‘10 / 2‘).

Specifies the list of nodes to be offset.

If *ON*, the offset will occur in the opposite direction indicated by the direction vector.

Edit Method



When one of these attributes is selected, the appropriate information to modify appears on the form. When this form is initially opened, none of the attributes are selected, therefore, only the Nodal Attributes box is displayed.

Used to input new node ID numbers for the listed nodes. See **Output ID List** (p. 25) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

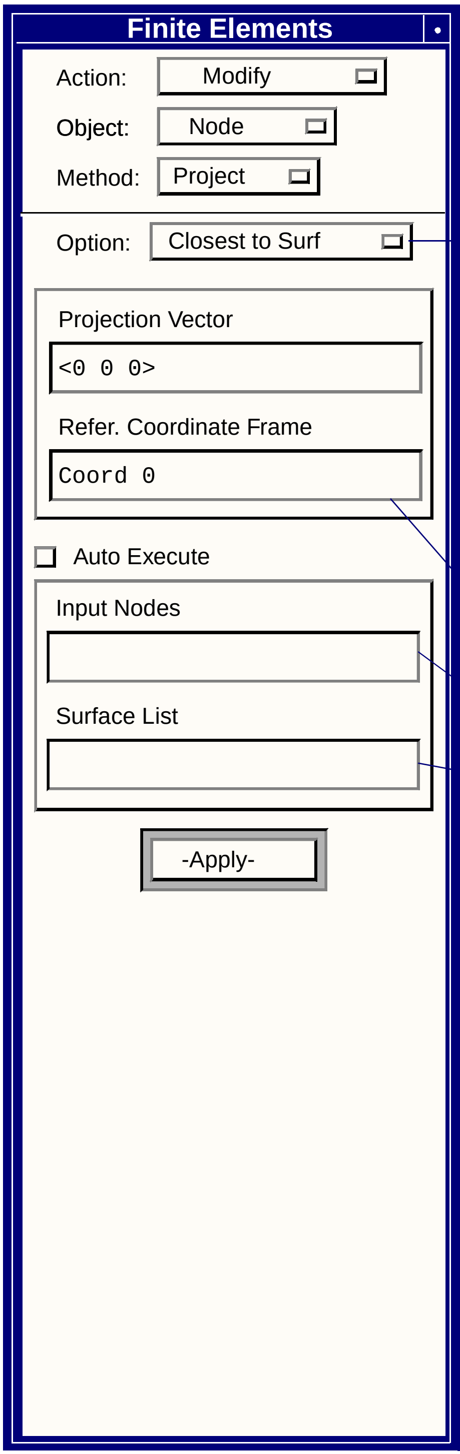
ID number of a coordinate frame where the displacements, degrees-of-freedom, constraints, and solution vectors are defined for the listed nodes.

ID number of a coordinate frame where the location is defined for the listed nodes.

New physical location(s) of the listed node(s). Enter coordinates in the global rectangular system or use the select menu.

Appears when any nodal attribute is selected. It lists the ID numbers of the nodes which are to be changed.

Project Method



Closest to Surf option will project the existing nodes by using the closest approach to the specified surface or face.

Define Vector option allows you to specify the coordinates of the Projection Vector and the Refer. Coordinate Frame to express the vector within. (**Example:** <1 1 0>). The Vector select menu will appear to allow you alternate ways to cursor define the vector direction.

View Vector option will project the existing points by using the view angle of the current viewport. MSC.Patran will project the existing nodes using the normal direction of the screen.

Closest to Curve option will project the existing nodes by using the closest approach to the specified curve or edge.

To Plane option will project the existing nodes by changing the selected coordinate value in a specified reference coordinate system.

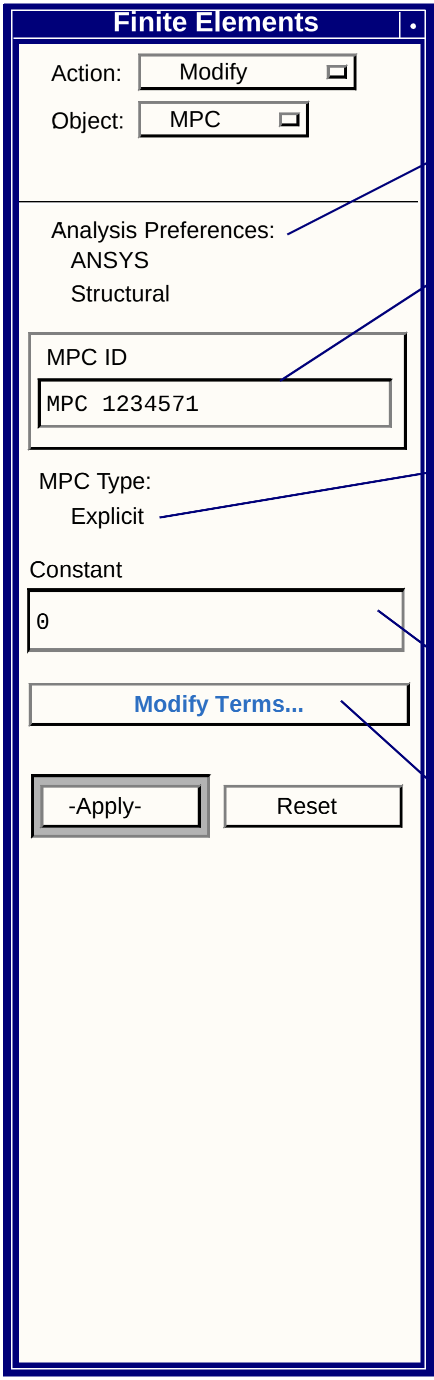
Projection Vector and Refer. Coordinate Frame is used if the Define Vector option is chosen.

Select Nodes to be projected with the selected option.

Specify Surfaces or Curves that the nodes will be projected onto.

Modifying MPCs

When Modify is the selected Action and MPC is the selected Object, the Modify MPC form is displayed. Use this form to modify the attributes of existing MPCs.



Indicates the current settings of the Analysis Code and Analysis Type Preferences.

Used to specify the MPC to be modified. The form will be updated automatically whenever the contents of this databox are changed to reflect the attributes of the existing MPC. If the MPC does not exist, the remaining widgets in this form are not displayed.

Indicates the MPC type currently selected. If the MPC is not valid for the current Analysis Code and Analysis Type preferences, this label will read:

Not Valid for Current Preferences

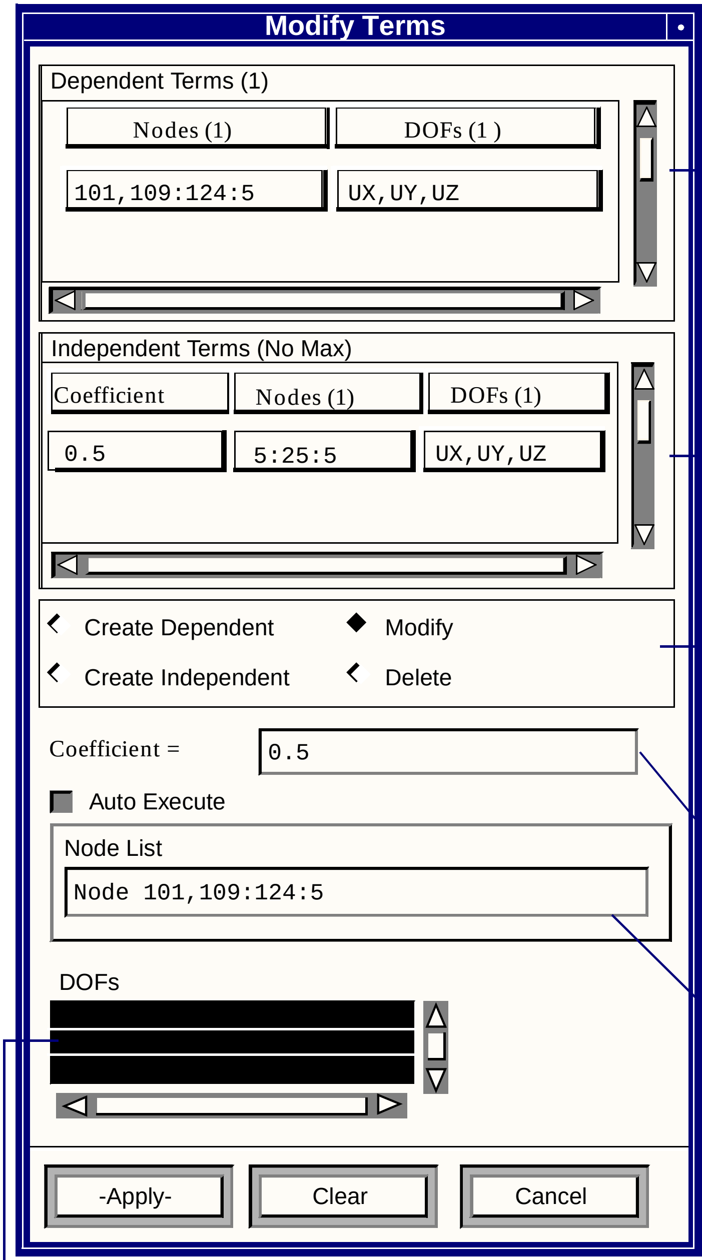
In this case, the remaining widgets will not be displayed.

Specifies a constant term, if supported by the current selected MPC type.

Brings up the *Modify Terms* form. Use to create, modify or delete dependent and independent terms.

Modify Terms

This form appears when the Modify Terms button is selected on the Modify MPC form. Use this form to modify the dependent and independent terms of a selected MPC.



Holds the dependent and independent term information as rows in the spreadsheet. The number of terms required is displayed in parentheses next to the spreadsheet label. A term consists of one or more of the following:

1. A sequencer number (not shown).
2. A nonzero coefficient.
3. A list of nodes (the required number is displayed in parentheses).
4. A list of degrees-of-freedom (the required number is listed in parentheses).

Existing terms can be selected for modification and deletion.

Sets the mode of the Apply function to:

1. Create a dependent term.
2. Create an independent term.
3. Modify a term, or
4. Delete a term.

The Create Dependent and Create Independent items are disabled once the maximum number of dependent or independent terms are created.

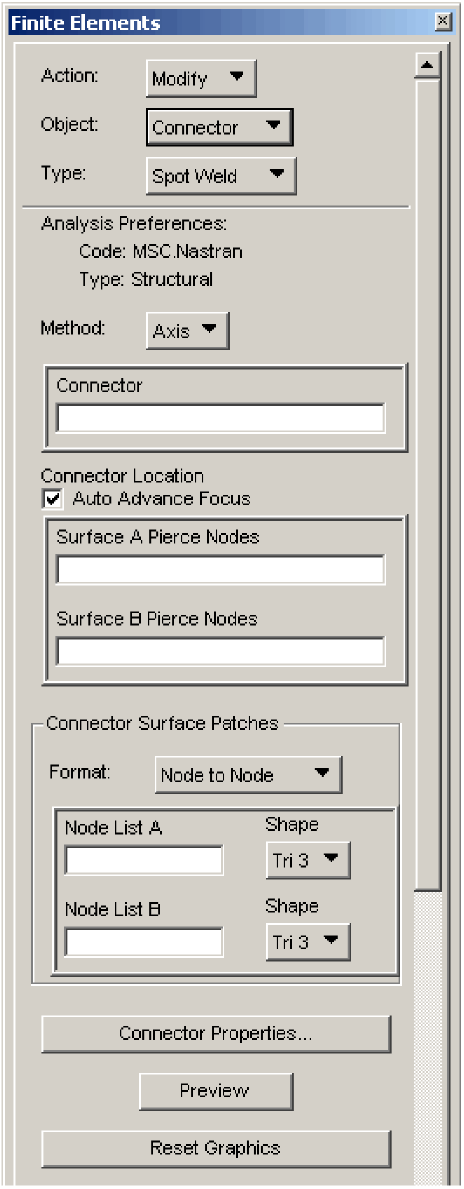
Specifies a *nonzero coefficient* for a term. This widget is displayed when creating or modifying a term which includes a Coefficient column.

Specifies the *nodes* for a term. This widget is displayed when creating or modifying a term which includes a Nodes column.

Select the *degrees-of-freedom* for a term. This widget is displayed when creating or modifying a term, which includes a DOFs column.

Modifying Spot Weld Connectors

The form under the Modify action is almost identical to the Create form, except the "Connector ID List" databox becomes a select databox. Connectors may then be selected from the screen, and the values within the form are automatically populated with the values for the selected connector, as are the values for the Connector Properties form.



Connector A single connector may be selected from the screen, or entered from the keyboard. For keyboard entry, the values are not populated until focus leaves the Connector select databox.

CHAPTER 14

The Delete Action

- Delete Action
- Delete Forms

14.1 Delete Action

The Delete action provides the capability to remove finite element entities from the model database. Submenus are provided to selectively delete any combination of finite element entities or specifically Node, Element, Mesh Seed definitions, Mesh on Curve/Surface/Solid, or MPC entities. By default, Auto Execute is selected which means MSC.Patran will automatically delete after the entities are selected.

If there are many finite element entities to be deleted, a percent complete form will show the status of the delete process for each entity type. When deletion is complete, a report appears in the command line indicating the number and IDs of the entities deleted, and the number and IDs of the entities not found and therefore not deleted.

The association of the deleted entity with other related entities is broken during deletion. Nodes, element properties, loads and boundary conditions, results and groups may become unreferenced due to deletion. Toggles are provided to delete unreferenced nodes and empty groups due to the delete function. The current group will not be deleted even if it becomes empty.

The Abort key may be selected at any time to halt the delete process, and the Undo button may be used to restore the deleted entities.

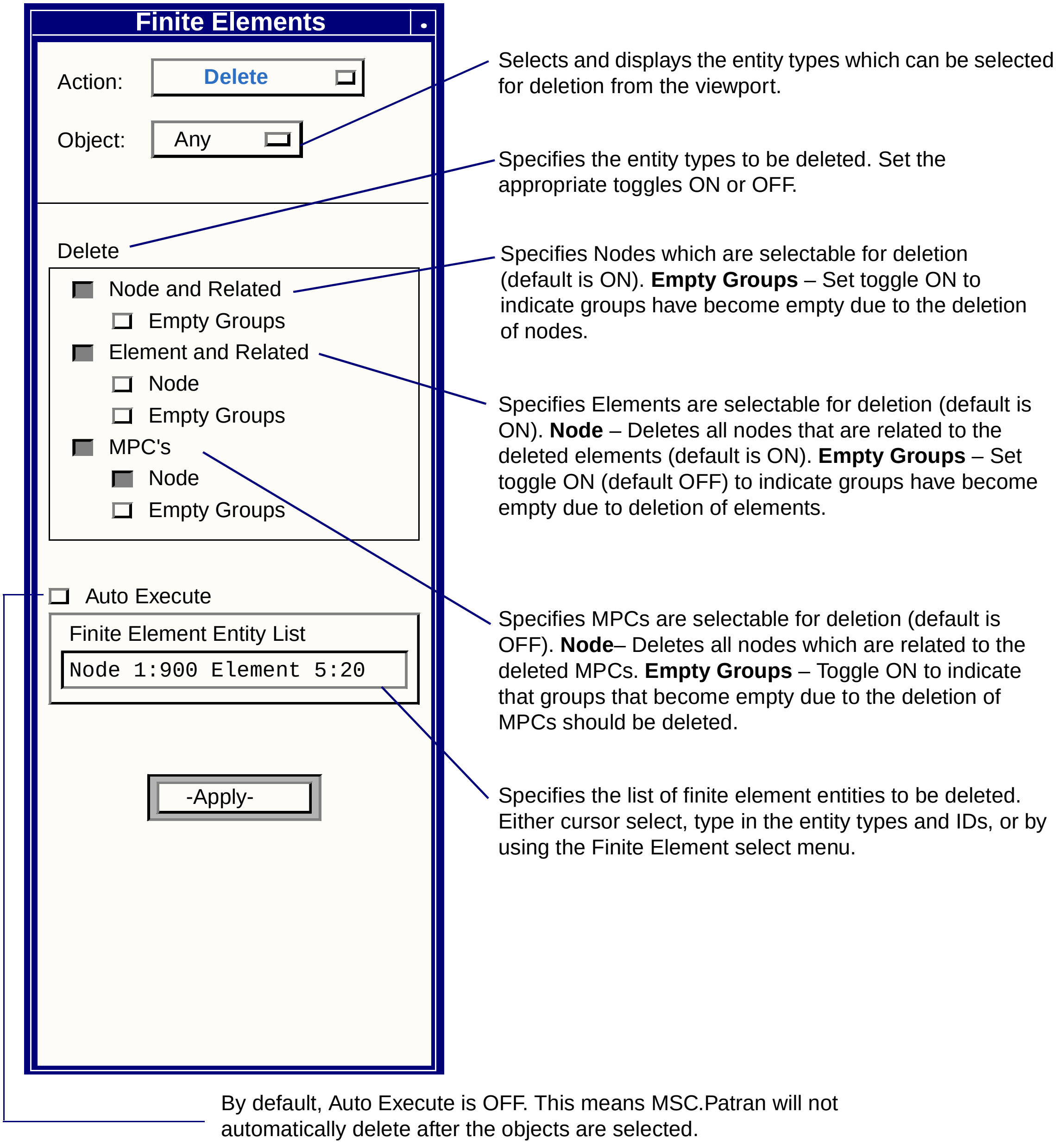
14.2 Delete Forms

When *Delete* is the selected action, the following options are available.

Object	Type	Description
Any		Allows for deletion of multiple types of finite element entities at once. Related nodes, element properties, load and boundary conditions, results and groups may become unreferenced due to deletion.
Mesh Seed		Deletes the mesh seed definitions from the specified edges.
Mesh Control		Deletes the mesh control applied to a surface.
Mesh	**Surface**	Deletes the mesh from the specified surfaces.
	Curve	Deletes the mesh from the specified curves.
	Solid	Deletes the mesh from the specified solids.
Node		Deletes the specified nodes. Element corner nodes will not be deleted. Related load and boundary conditions, results are disassociated with the deleted nodes but they are not deleted. Any nodes associated with a DOF list will be removed from the nodes portion of the DOF list term. A toggle is provided to delete empty groups due to the deletion.
Element		Deletes the specified elements. Nodes, element properties, load and boundary conditions, results and groups are disassociated with the deleted elements but they are not deleted. A toggle is provided to delete related nodes and empty groups due to the deletion.
MPC		Deletes the multi-point constraints. Nodes and groups are disassociated with the deleted MPCs but they are not deleted. A toggle is provided to delete related nodes and empty groups due to the deletion.
Connector		Deletes the connectors.
Superelement		Deletes the superelements.
DOF List		Deletes the specified degree-of-freedom (DOF) lists.

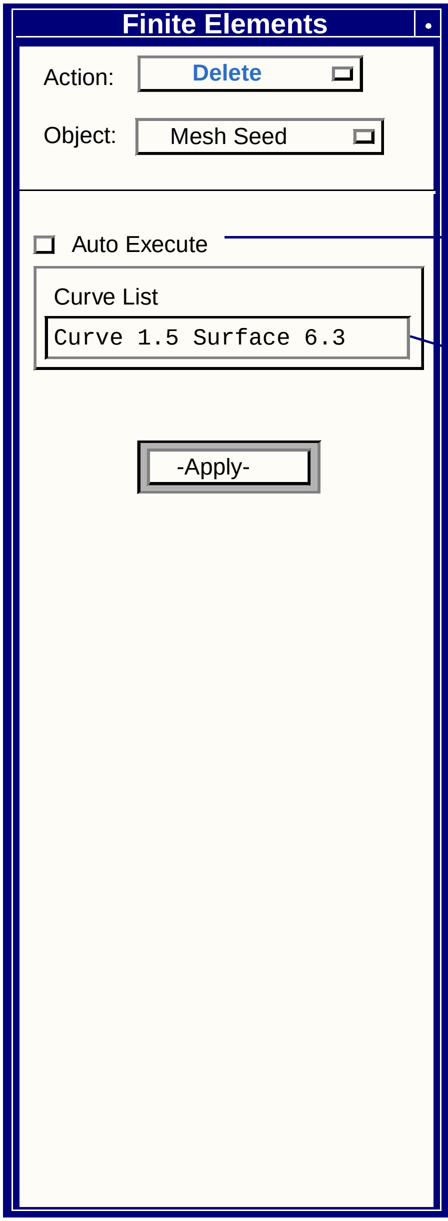
Delete - Any

Use this form to delete multiple types of finite element entities at one time. Any combination of elements, nodes, and multi-point constraints may be selected for deletion. When deleting elements and nodes, the mesh on curves, surfaces and solids may also be deleted. However, mesh seeds can only be deleted through the Delete / Mesh Seed menu. Nodes, element properties, loads and boundary conditions, results and groups may become unreferenced due to deletion. Toggles are provided to delete unreferenced nodes and empty groups due to the delete operation.



Delete - Mesh Seed

Use this form to delete an existing mesh seed definition for a list of specified edges. The edges may be curves, or edges of surfaces or solids. When deletion is complete, a report appears in the command line indicating the number and IDs of the edges which were deleted.



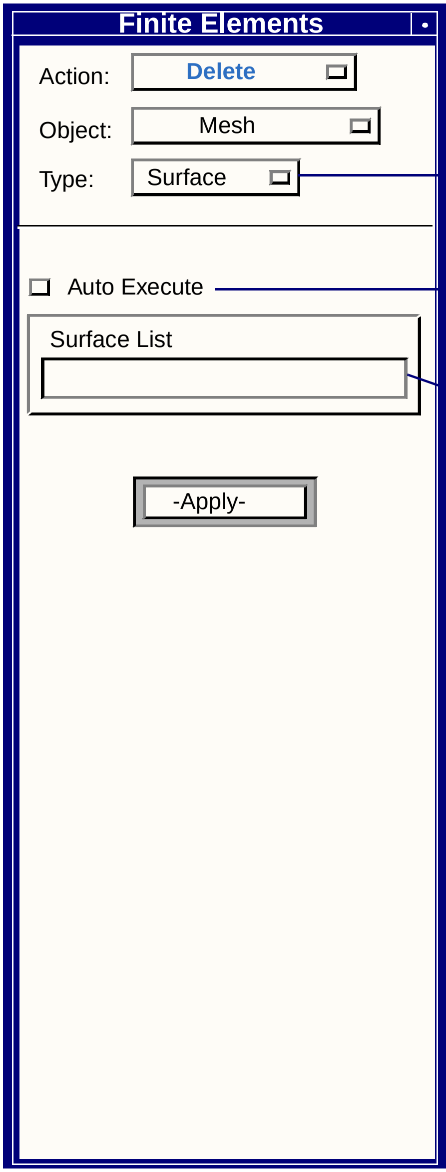
By default, Auto Execute is OFF. This means MSC.Patran will not automatically delete after the objects are selected.

Specifies the list of edges containing the mesh seed definitions to be deleted. Either cursor select the existing curves or edges of surfaces or solids or specify the edge entity type and IDs. (**Example:** Curve 10, Surface 12.1, Solid 22.5.2.)

The abort key may be pressed at any time to halt the delete process, and the Undo button may be used to restore the deleted mesh seed definitions to their respective edges.

Delete - Mesh (Surface)

Use this form to delete an existing mesh of nodes and elements applied to one or more surfaces, or solid faces. When deletion is complete, a report appears in the command line indicating the number and IDs of the entities from which meshes were deleted.



Types which can be selected are: Curve, Surface, or Solid.

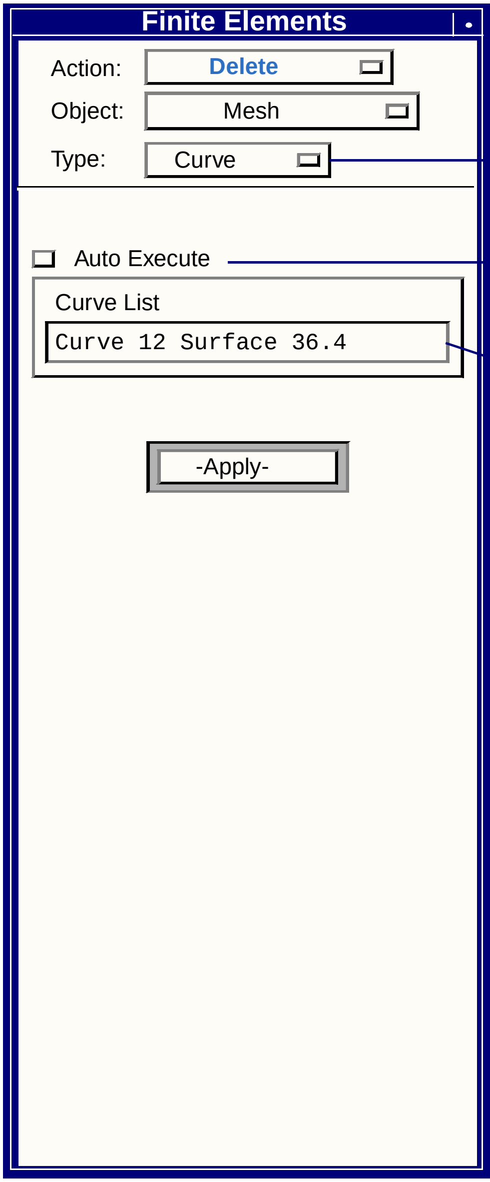
By default, Auto Execute is OFF. This means MSC.Patran will not automatically delete after the objects are selected.

Specifies the list of geometric entities that are meshed by cursor or by entering the entity type and IDs. (**Example**, to delete a mesh from a Surface enter: Surface 12, Solid 22.4.)

The abort key may be pressed at any time to halt the delete process, and the Undo button may be used to restore the deleted mesh of nodes and elements.

Delete - Mesh (Curve)

Use this form to delete an existing mesh of nodes and elements applied to one or more curves, or surface or solid edges.



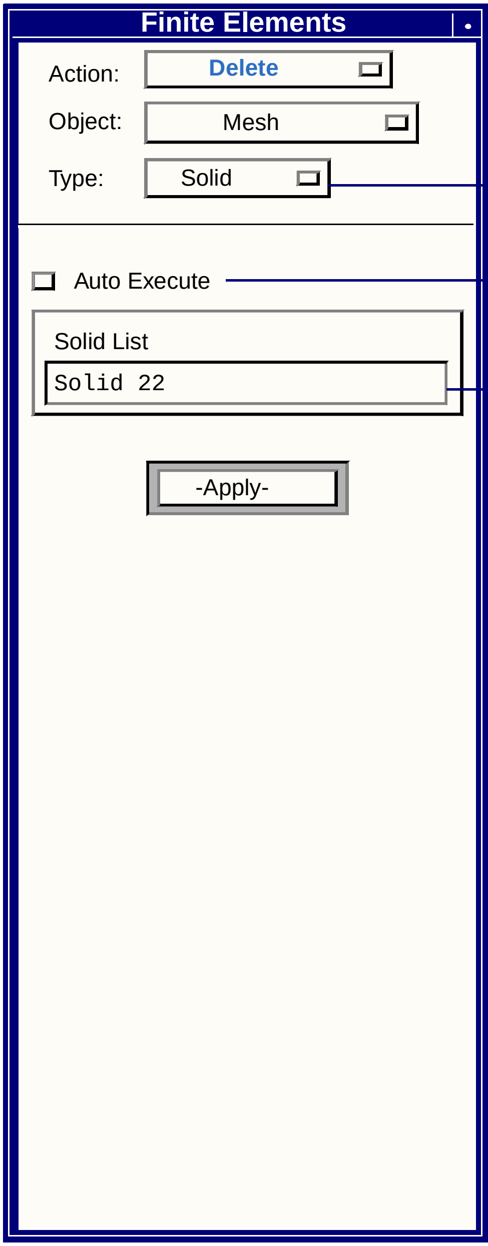
Defines the general type of mesh to be deleted. This can be set to Curve, Surface and Solid.

By default, Auto Execute is OFF. This means MSC.Patran will not automatically delete a mesh after each geometric entity is selected.

Specifies the list of geometric entities. Either cursor select existing meshed entities, use the left mouse button, or enter the IDs. (**Example:** Curve 12, Surface 30.4.)

Delete - Mesh (Solid)

Use this form to delete an existing mesh of nodes and elements applied to one or more solids.

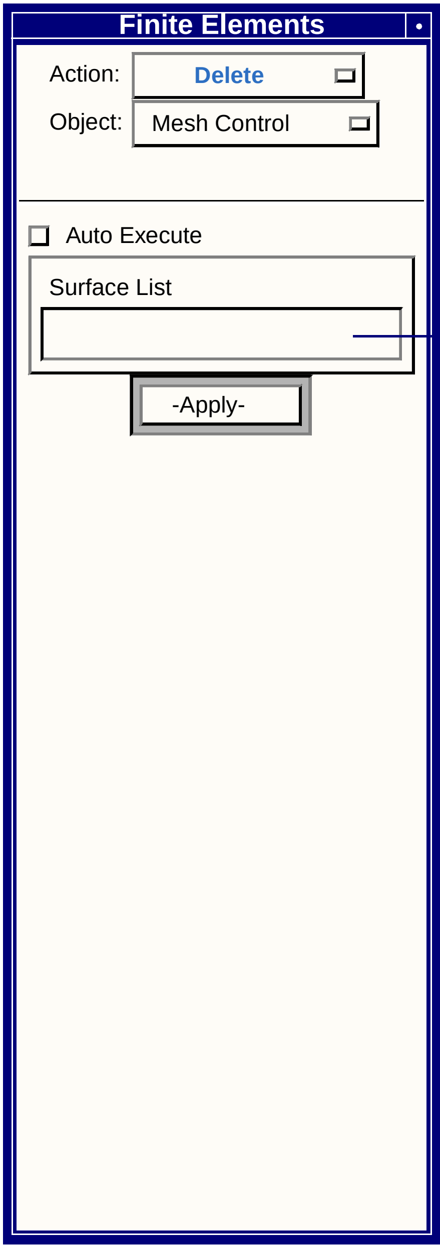


Defines the general type of mesh to be deleted. This can be set to Curve, Surface and Solid.

By default, Auto Execute is OFF. This means MSC.Patran will not automatically delete a mesh after each geometric entity is selected.

Specifies the list of geometric entities. Either cursor select existing meshed entities, use the left mouse button, or enter the IDs. (**Example:** Solid 22.)

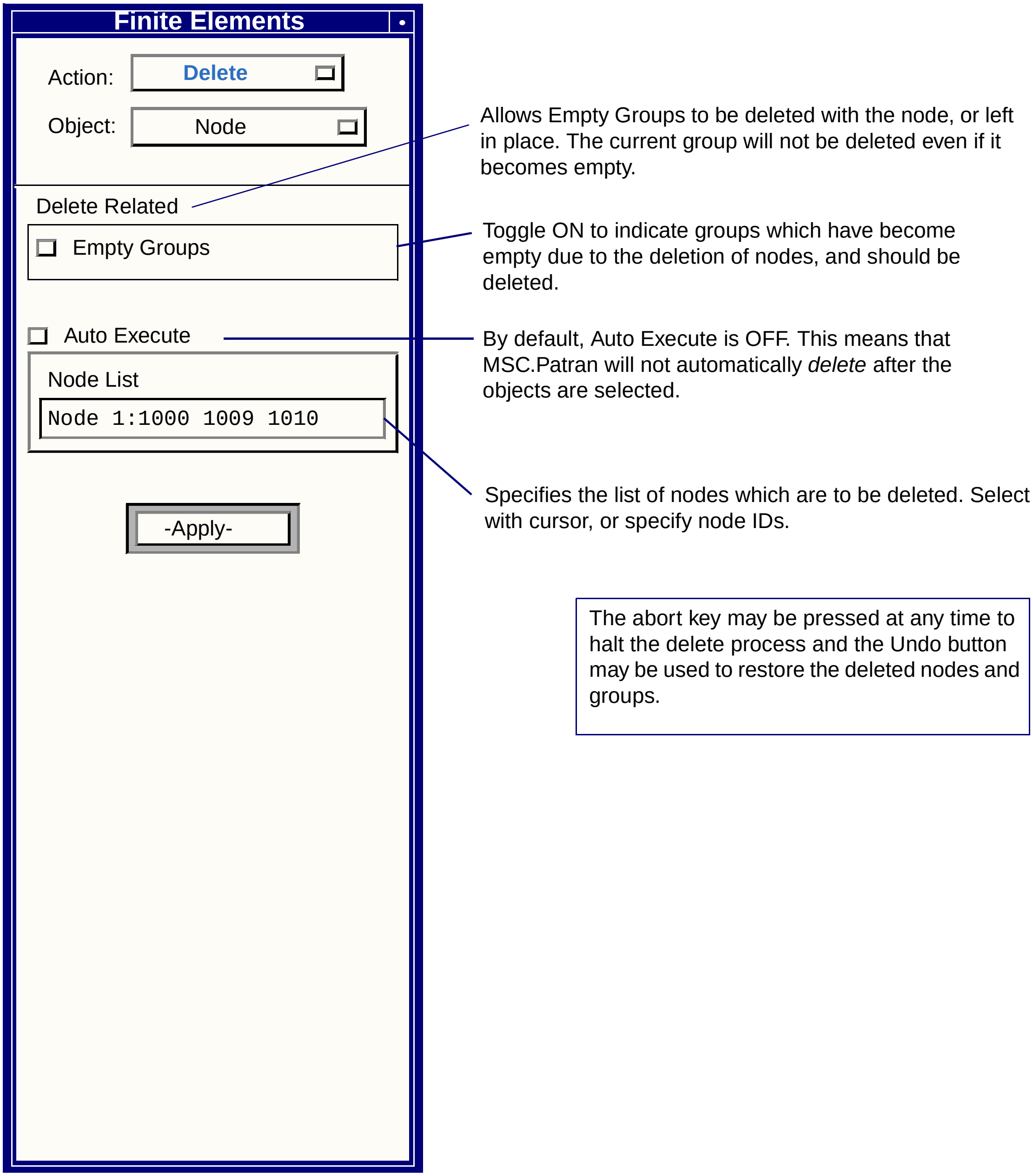
Delete - Mesh Control



List of surfaces from which mesh control should be deleted.

Delete - Node

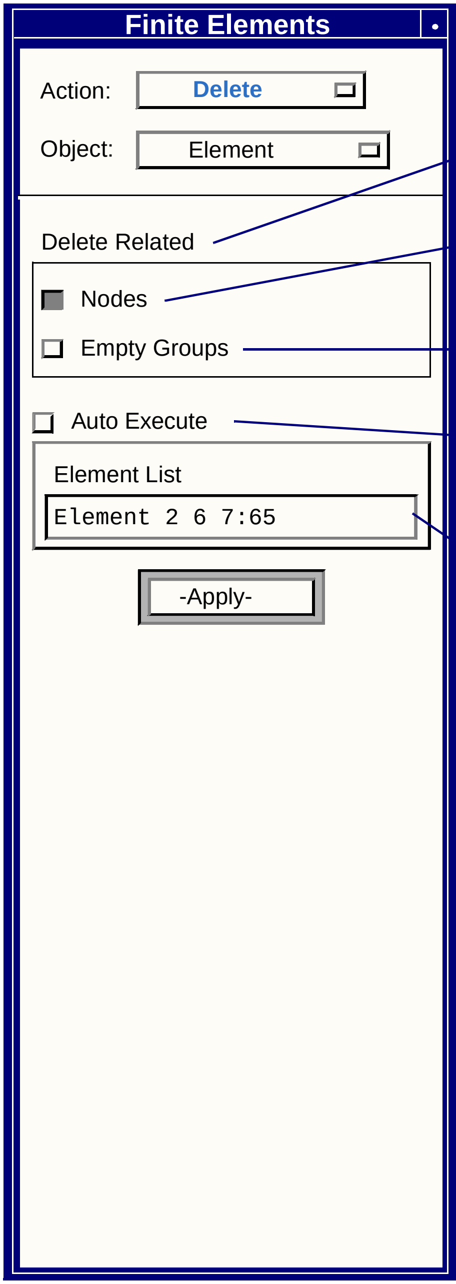
Use this form to delete existing nodes from the model database. Element corner nodes will not be deleted. Related loads and boundary conditions, results and groups are disassociated with the deleted nodes but they are not deleted. Any nodes associated with a DOF list will be removed from the nodes portion of the DOF list term. A toggle is provided to delete groups that become empty due to the deletion of the nodes. When deletion is complete a report appears in the command line indicating the number and IDs of the nodes deleted.



The abort key may be pressed at any time to halt the delete process and the Undo button may be used to restore the deleted nodes and groups.

Delete - Element

Use this form to delete existing elements from the model database. Related nodes, element properties, loads and boundary conditions, results and groups are disassociated from the deleted elements, but they are not deleted. A toggle is provided to delete all related nodes and empty groups due to the deletion of elements.



Allows for the related nodes and Empty groups to be deleted with the element or left alone. The current group will not be deleted even if it becomes empty.

Deletes all nodes that are related to the elements which have been deleted (default is ON).

Toggle ON to indicate groups which have become empty due to the deletion of elements, and should be deleted.

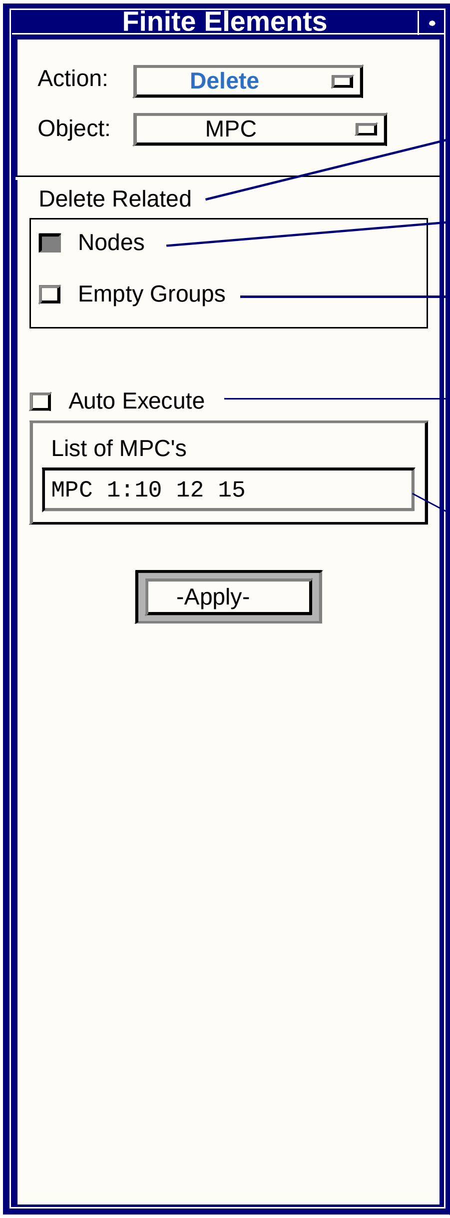
By default, Auto Execute is OFF. This means that MSC.Patran will not automatically delete after the objects are selected.

Specifies the list of elements. Either cursor select, or specifies element IDs. Through the element select menu, elements may be deleted by type (for instance only bar elements or triangular elements... etc.).

The abort key may be pressed at any time to halt the delete process, and the Undo button may be used to restore the deleted elements and related nodes and groups.

Delete - MPC

Use this form to delete an existing multi-point constraint (MPC) from the database. Related nodes and groups are disassociated from the deleted MPCs, but they are not deleted. A toggle is provided to delete all related nodes and empty groups due to the deletion of the MPCs.



Allows the related nodes and empty groups to be deleted or left in place.

Deletes all nodes which are related to the MPCs that have been deleted (default is ON).

Toggle ON to indicate groups which have become empty due to the deletion of MPCs and should be deleted.

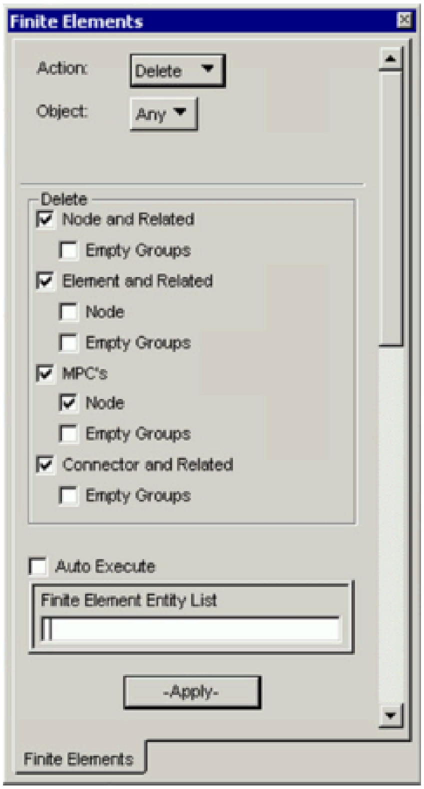
By default, Auto Execute is OFF. This means that MSC.Patran will not automatically delete after the objects are selected.

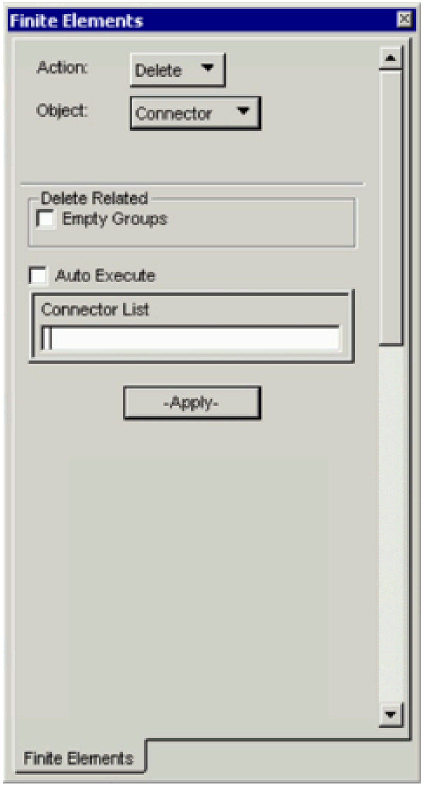
Specifies the list of MPCs to be deleted. Select with the cursor or type in the MPC IDs.

The abort key may be selected at any time to halt the delete process, and the Undo button may be used to restore the deleted MPCs and related nodes and groups.

Delete - Connector

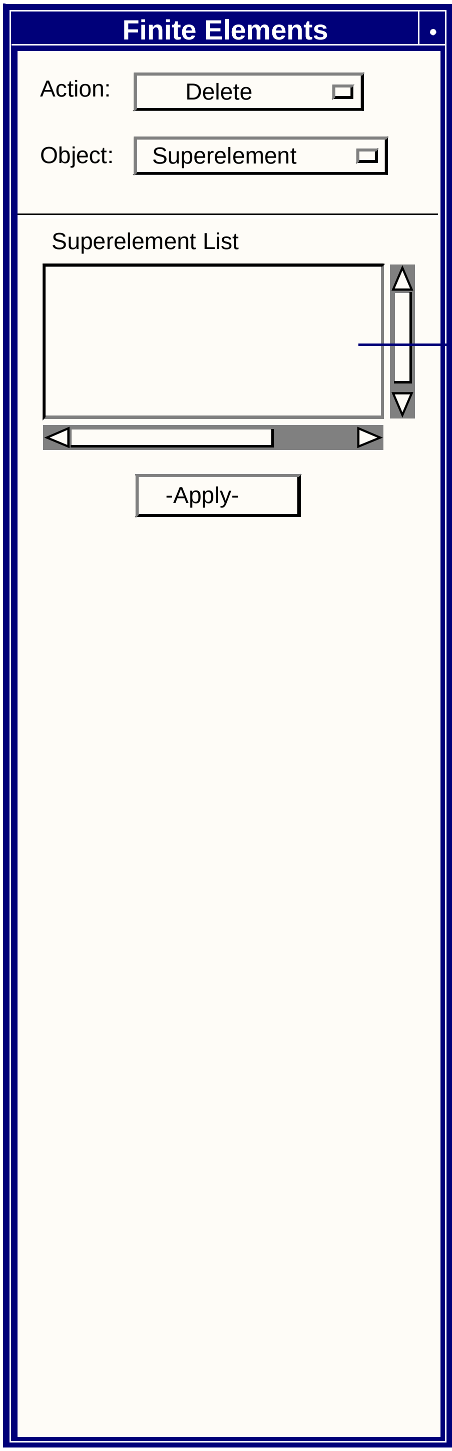
Use this form to delete a connector from the database. Related nodes and groups are disassociated from the deleted connector, but they are not deleted. A toggle is provided to delete all related nodes and empty groups due to the deletion of the connector.





Delete - Superelement

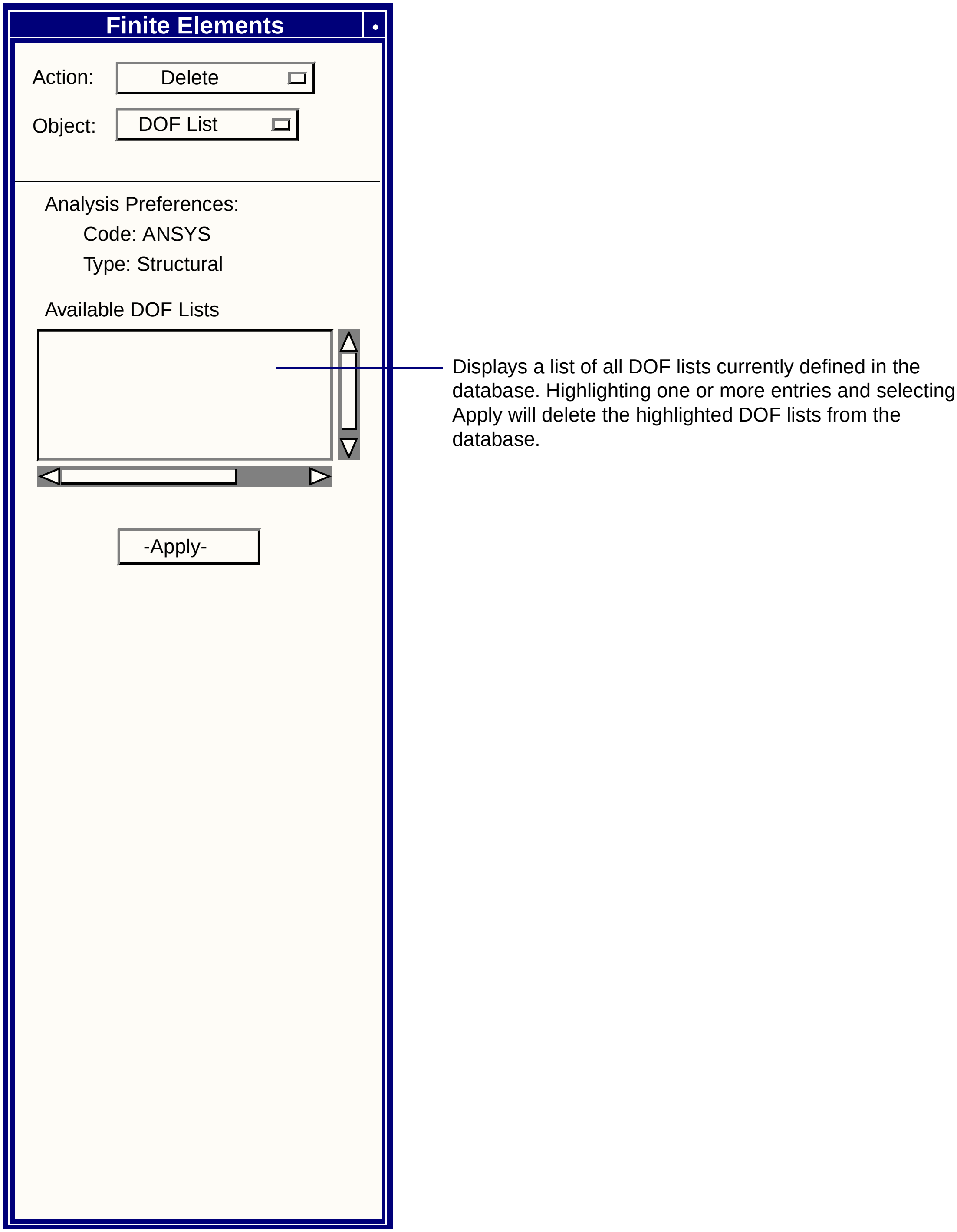
Use this form to delete superelements from the database. Note that this is currently available only for the MSC.Nastran analysis preference.



List of existing superelements. Only the highlighted superelements are deleted.

Delete - DOF List

Use this form to delete degree-of-freedom (DOF) lists from the database. Note that this is currently available only for the ANSYS and ANSYS 5 analysis preference.



CHAPTER

15

The MSC.Patran Element Library

- Introduction
- Beam Element Topology
- Tria Element Topology
- Quad Element Topology
- Tetrahedral Element Topology
- Wedge Element Topology
- Hex Element Topology
- MSC.Patran's Element Library

15.1 Introduction

The MSC.Patran template database file, `template.db`, contains a “generic” set of finite element topologies. By default, when opening a new database, the element topology library is included. Topology, in the context of a finite element, is the relative node, edge and face numbering scheme for each element of the same topology. The MSC.Patran library is compatible with earlier versions of MSC.Patran (PATRAN Release 2.5). MSC.Patran also provides additional information about each element topology which was not available in the earlier MSC.Patran versions:

- Nodal parametric locations
- Edge numbering
- Face numbering
- Face sense
- Corresponding degenerate element topology ID

Where possible, the ISO 10303-104, Application Resources: Finite Element Analysis document, which is part of International Standard ISO 10303-Product Data Representation and Exchange (STEP), was used to define the element topologies. If the ISO standard was found to be in conflict with earlier versions of MSC.Patran, the MSC.Patran convention took precedence. The ISO standard for numbering edges and faces of elements is used.

Face and edge numbering are important for assigning element attributes, such as pressures applied to a solid element face. In MSC.Patran, you may select an edge or a face of an element with the cursor. An example of the syntax, used in the Select Databox to describe an edge of hex element 1, would be **`elem 1.2.3`**, which refers to edge 3 of face 2 of element 1.

The element topology tables listed in sections 13.2 through 13.7 are used to construct and interpret the syntax of the Select Databox string. **MSC.Patran’s Element Library** (p. 378) provides illustrations of each element type and topology, and their node locations.

Important:	The face sense is interpreted as positive if the normal is pointing away from (towards the outside) the element, using the right hand rule. This only applies to volume elements (element dimensionality = 3).

Parametric coordinate systems

Rectangular. [Xi/Eta/Zeta] is used for Tet/Wedge/Hex elements. Values can either have a range of -1 to 1 or 0 to 1 depending on the case where an area or volume coordinate systems can apply (Tet/Wedge elements). [Xi/Eta] applies to a Tri or Quad element. Values range from 0 to 1 for the Tri, and -1 to 1 for the Quad. [Xi] applies to a Bar element. Values range from -1 to 1.

Area. [L1/L2/L3] is used for locating a point within a triangular area. Values range from 0 to 1, and the sum of all cordinates is equal to 1. The values correspond to the weighting with respect to the 3 corners of a triangle. For a Tri or Wedge element which will use [Xi/Eta] and [Xi/Eta/Zeta], the Xi/Eta value will range from 0 to 1, and we can determine L1/L2/L3 as :

$$L1 = 1.0 - Xi - Eta$$

$$L2 = Xi$$

$$L3 = Eta$$

Volume. [L1/L2/L3/L4] is used for locating a point within a tetrahedral volume. Values range from 0 to 1, and the sum of all cordinates is equal to 1. The values correspond to the weighting with respect to the 4 corners of a tetrahedron. For a Tet element which will use [Xi/Eta/Zeta], the Xi/Eta/Zeta value will range from 0 to 1, and we can determine L1/L2/L3/L4 as :

L1 = 1.0 - Xi - Eta - Zeta

L2 = Xi

L3 = Eta

L4 = Zeta

15.2 Beam Element Topology

MSC.Patran contains three different beam element topologies: Bar2, Bar3 and Bar4.

General Data

Shape = Beam

Element dimensionality= 1

Number of corner nodes = 2

Number of edges = 1

Number of faces = 0

Number of face edges = 0

Specific Data - Bar2

Element name = Bar2

Number of nodes = 2

Order = linear

Degenerate element name = <none>

Table 15-1 Bar2 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2

Table 15-2 Bar2 Node Parametric Coordinates

Node Number	Xi
1	-1.0
2	1.0

For more information, see **Bar2** (p. 378).

Specific Data - Bar3

Element name = Bar3

Number of nodes = 3

Order = Quadratic

Degenerate element name = <none>

Table 15-3 Bar3 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	3	2

Table 15-4 Bar3 Node Parametric Coordinates

Node Number	Xi
1	-1.0
2	1.0
3	0.0

For more information, see **Bar3** (p. 380).

Specific Data - Bar4

Element name = Bar4

Number of nodes = 4

Order = Cubic

Degenerate element name = <none>

Table 15-5 Bar4 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	3	4	2

Table 15-6 Bar4 Node Parametric Coordinates

Node Number	Xi
1	-1.0
2	1.0
3	-1/3
4	1/3

For more information, see **Bar4** (p. 388).

15.3 Tria Element Topology

MSC.Patran contains six different triangular element topologies: Tria3, Tria4, Tria6, Tria7, Tria9, Tria13.

General Shape

For Tri elements, area coordinates [L1/L2/L3] are commonly used. See **Area** (p. 318) coordinate system for more information.

Tri elements can be obtained by degenerating a Quad element.

1. Quad corner node 2 collapses onto 1.
2. Tri corner nodes 1/2/3 match 1/3/4 for the Quad.

General Data

Shape = Triangular

Element dimensionality= 2

Number of corner nodes = 3

Number of edges = 3

Number of faces = 1

Number of face edges = 3

Table 15-7 Tria Face Numbering

Face ID	Sense	Edge 1	Edge 2	Edge 3
1	1	1	2	3

Specific Data - Tria3

Element name = Tria3

Number of nodes = 3

Order = linear

Degenerate element name = <none>

Table 15-8 Tria3 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	3
3	3	1

Table 15-9 Tria3 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
1	0.0, 0.0
2	1.0, 0.0,0.0
3	0.0, 1.0

To obtain a Tri3 by degenerating a Quad4, the following are corresponding nodes:

Tri3	Quad4
1	1
2	3
3	4

For more information, see **Tri3** (p. 378).

Specific Data - Tria4

Element name = Tria4

Number of nodes = 4

Order = linear

Degenerate element name = <none>

Table 15-10 Tria4 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	3
3	3	1

Table 15-11 Tria4 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
1	0.0, 0.0
2	1.0, 0.0

Table 15-11 Tria4 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
3	0.0, 1.0
4	1/3, 1/3

To obtain a Tri4 by degenerating a Quad5, the following are corresponding nodes:

Tri4	Quad5
1	1
2	3
3	4
4	5

For more information, see **Tri4** (p. 379).

Specific Data - Tria6

Element name = Tria6

Number of nodes = 6

Order = Quadratic

Degenerate element name = <none>

Table 15-12 Tria6 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	4	2
2	2	5	3
3	3	6	1

Table 15-13 Tria6 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
1	0.0, 0.0
2	1.0, 0.0
3	0.0, 1.0
4	0.5, 0.0

Table 15-13 Tria6 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
5	0.5, 0.5
6	0.0, 0.5

To obtain a Tri6 by degenerating a Quad8, the following are corresponding nodes:

Tri6	Quad8
1	1
2	3
3	4
4	6
5	7
6	8

For more information, see **Tri6** (p. 380).

Specific Data - Tria7

Element name = Tria7

Number of nodes = 7

Order = Quadratic

Degenerate element name = <none>

Table 15-14 Tria7 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	4	2
2	2	5	3
3	3	6	1

Table 15-15 Tria7 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
1	0.0, 0.0
2	1.0, 0.0
3	0.0, 1.0
4	0.5, 0.0

Table 15-15 Tria7 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
5	0.5, 0.5
6	0.0, 0.5
7	1/3, 1/3

To obtain a Tri7 by degenerating a Quad9, the following are corresponding nodes:

Tri7	Quad9
1	1
2	3
3	4
4	6
5	7
6	8
7	9

For more information, see **Tri7** (p. 381).

Specific Data - Tria9

Element name = Tria9

Number of nodes = 9

Order = Cubic

Degenerate element name = <none>

Table 15-16 Tria9 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	4	5	2
2	2	6	7	3
3	3	8	9	1

Table 15-17 Tria9 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
1	0.0, 0.0
2	1.0, 0.0
3	0.0, 1.0

Table 15-17 Tria9 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
4	1/3, 0.0
5	2/3, 0.0
6	1/3, 2/3
7	1/3, 2/3
8	0.0, 2/3
9	0.0, 1/3

To obtain a Tri9 by degenerating a Quad12, the following are corresponding nodes:

Tri9	Quad12
1	1
2	3
3	4
4	7
5	8
6	9
7	10
8	11
9	12

For more information, see **Tri9** (p. 388).

Specific Data - Tria13

Element name = Tria13

Number of nodes = 13

Order = Cubic

Degenerate element name = <none>

Table 15-18 Tria13 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	4	5	2
2	2	6	7	3
3	3	8	9	1

Table 15-19 Tria13 Node Parametric Coordinates

Node Number	Xi/Eta or L2/L3
1	0.0, 0.0
2	1.0, 0.0
3	0.0, 1.0
4	1/3, 0.0
5	2/3, 0.0
6	2/3, 1/3
7	1/3, 2/3
8	0.0, 2/3
9	0.0, 1/3
10	2/9, 1/9
11	4/9, 2/9
12	2/9, 4/9
13	1/9, 2/9

To obtain a Tri13 by degenerating a Quad16, the following are corresponding nodes:

Tri13	Quad16
1	1
2	3
3	4
4	7
5	8
6	9
7	10
8	11
9	12
10	14
11	15
12	16
13	13

For more information, see **Tri13** (p. 389).

15.4 Quad Element Topology

MSC.Patran contains six different quadrilateral element topologies: Quad4, Quad5, Quad8, Quad9, Quad12, Quad16.

General Data

Shape = Quadrilateral

Element dimensionality= 2

Number of corner nodes = 4

Number of edges = 4

Number of faces = 1

Number of face edges = 4

Table 15-20 Quad Face Numbering

Face ID	Sense	Edge 1	Edge 2	Edge 3	Edge 4
1	1	1	2	3	4

Specific Data - Quad4

Element name = Quad4

Number of nodes = 4

Order = linear

Degenerate element name = Tria3

Table 15-21 Quad4 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	3
3	3	4
4	4	1

Table 15-22 Quad4 Node Parametric Coordinates

Node Number	Xi/Eta
1	-1.0, -1.0
2	1.0, -1.0
3	1.0, 1.0
4	-1.0, 1.0

For more information, see Quad4 (p. 378).

Specific Data - Quad5

Element name = Quad5

Number of nodes = 5

Order = linear

Degenerate element name = Tria4

Table 15-23 Quad5 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	3
3	3	4
4	4	1

Table 15-24 Quad5 Node Parametric Coordinates

Node Number	Xi/Eta
1	-1.0, -1.0
2	1.0, -1.0
3	1.0, 1.0
4	-1.0, 1.0
5	0.0, 0.0

For more information, see Quad5 (p. 379).

Specific Data - Quad8

Element name = Quad8

Number of nodes = 8

Order = Quadratic

Degenerate element name = Tria6

Table 15-25 Quad8 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	5	2
2	2	6	3
3	3	7	4
4	4	8	1

Table 15-26 Quad8 Node Parametric Coordinates

Node Number	Xi/Eta
1	-1.0, -1.0
2	1.0, -1.0
3	1.0, 1.0
4	-1.0, 1.0
5	0.0, -1.0
6	1.0, 0.0
7	0.0, 1.0
8	-1.0, 0.0

For more information, see Quad8 (p. 380).

Specific Data - Quad9

Element name = Quad9

Number of nodes = 9

Order = Quadratic

Degenerate element name = Tria7

Table 15-27 Quad9 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	5	2
2	2	6	3
3	3	7	4
4	4	8	1

Table 15-28 Quad9 Node Parametric Coordinates

Node Number	Xi/Eta
1	-1.0, -1.0
2	1.0, -1.0
3	1.0, 1.0
4	-1.0, 1.0
5	0.0, -1.0
6	1.0, 0.0
7	0.0, 1.0
8	-1.0, 0.0
9	0.0, 0.0

For more information, see Quad9 (p. 381).

Specific Data - Quad12

Element name = Quad12

Number of nodes = 12

Order = Cubic

Degenerate element name = Tria9

Table 15-29 Quad12 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	5	6	2
2	2	7	8	3

Table 15-29 Quad12 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
3	3	9	10	4
4	4	11	12	1

Table 15-30 Quad12 Node Parametric Coordinates

Node Number	Xi/Eta
1	-1.0, -1.0
2	1.0, -1.0
3	1.0, 1.0
4	-1.0, 1.0
5	1/3, -1.0
6	1/3, -1.0
7	1.0,-1/3
8	1.0,1/3
9	1/3, 1.0
10	-1/3, 1.0
11	-1.0,1/3
12	-1.0,-1/3

For more information, see **Quad12** (p. 388).

Specific Data - Quad16

Element name = Quad16

Number of nodes = 16

Order = Cubic

Degenerate element name = Tria13

Table 15-31 Quad16 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	5	6	2
2	2	7	8	3

Table 15-31 Quad16 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
3	3	9	10	4
4	4	11	12	1

Table 15-32 Quad16 Node Parametric Coordinates

Node Number	Xi/Eta
1	-1.0, -1.0
2	1.0, -1.0
3	1.0, 1.0
4	-1.0, 1.0
5	-1 / 3, -1.0
6	1 / 3, -1.0
7	1.0,-1 / 3
8	1.0,1 / 3
9	1 / 3, 1.0
10	-1 / 3, 1.0
11	-1.0,1 / 3
12	-1.0,-1 / 3
13	-1 / 3,-1 / 3
14	1 / 3, -1 / 3
15	1 / 3, 1 / 3
16	-1 / 3, 1 / 3

For more information, see **Quad16** (p. 389).

15.5 Tetrahedral Element Topology

MSC.Patran contains eight different tetrahedral element topologies: Tet4, Tet5, Tet10, Tet11, Tet14, Tet15, Tet16, Tet40.

General Data

Shape = Tetrahedral

Element dimensionality= 3

Number of corner nodes = 4

Number of edges = 6

Number of faces = 4

Number of face edges = 3

General Shape

For Tet elements, volume coordinates [L1/L2/L3/L4] are commonly used. See **Volume** (p. 319) coordinate system for more information.

Tet elements can be obtained by degenerating Hex elements.

1. Hex corner nodes 2/3/4 collapse onto 1, and 6 collapses onto 5.
2. Tet corner nodes 1/2/3/4 match 1/5/7/8 for the Hex.

Table 15-33 Tetrahedral Face Numbering

Face ID	Sense	Edge 1	Edge 2	Edge 3
1	-1	1	2	3
2	1	1	5	4
3	1	2	6	5
4	1	3	4	6

Specific Data - Tet4

Element name = Tet4

Number of nodes = 4

Order = linear

Degenerate element name = <none>

Table 15-34 Tet4 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	3

Table 15-34 Tet4 Edge Numbering

Edge Number	Node 1	Node 2
3	3	1
4	1	4
5	2	4
6	3	4

Table 15-35 Tet4 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 0.0
4	0.0, 0.0, 1.0

To obtain a Tet4 by degenerating a Hex8, the following are corresponding nodes:

Tet4	Hex8
1	1
2	5
3	7
4	8

For more information, see Tet4 (p. 378).

Specific Data - Tet5

Element name = Tet5

Number of nodes = 5

Order = linear

Degenerate element name = <none>

Table 15-36 Tet5 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	3

Table 15-36 Tet5 Edge Numbering

Edge Number	Node 1	Node 2
3	3	1
4	1	4
5	2	4
6	3	4

Table 15-37 Tet5 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 1.0
4	0.0, 0.0, 1.0
5	1/4, 1/4, 1/4

To obtain a Tet5 by degenerating a Hex9, the following are corresponding nodes:

Tet5	Hex9
1	1
2	5
3	7
4	8
5	9

For more information, see **Tet5** (p. 379).

Specific Data - Tet10

Element name = Tet10

Number of nodes = 10

Order = Quadratic

Degenerate element name = <none>

Table 15-38 Tet10 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	5	2
2	2	6	3
3	3	7	1
4	1	8	4
5	2	9	4
6	3	10	4

Table 15-39 Tet10 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 0.0
4	0.0, 0.0, 1.0
5	0.5, 0.0, 0.0
6	0.5, 0.5, 0.0
7	0.0, 0.5, 0.0
8	0.0, 0.0, 0.5
9	0.5, 0.0, 0.5
10	0.0, 0.5, 0.5

To obtain a Tet10 by degenerating a Hex20, the following are corresponding nodes:

Tet10	Hex20
1	1
2	5
3	7
4	8
5	13
6	18
7	15
8	16

Tet10	Hex20
9	20
10	19

For more information, see Tet10 (p. 380).

Specific Data - Tet11

Element name = Tet11

Number of nodes = 11

Order = Quadratic

Degenerate element name = <none>

Table 15-40 Tet11 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	5	2
2	2	6	3
3	3	7	1
4	1	8	4
5	2	9	4
6	3	10	4

Table 15-41 Tet11 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 0.0
4	0.0, 0.0, 1.0
5	0.5, 0.0, 0.0
6	0.5, 0.5, 0.0
7	0.0, 0.5, 0.0
8	0.0, 0.0, 0.5
9	0.5, 0.0, 0.5
10	0.0, 0.5, 0.5
11	1/4, 1/4, 1/4

To obtain a Tet11 by degenerating a Hex21, the following are corresponding nodes:

Tet11	Hex21
1	1
2	5
3	7
4	8
5	13
6	18
7	15
8	16
9	20
10	19
11	21

For more information, see **Tet11** (p. 381).

Specific Data - Tet14

Element name = Tet14

Number of nodes = 14

Order = Quadratic

Degenerate element name = <none>

Table 15-42 Tet14 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	5	2
2	2	6	3
3	3	7	1
4	1	8	4
5	2	9	4
6	3	10	4

Table 15-43 Tet14 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 0.0
4	0.0, 0.0, 1.0
5	0.5, 0.0, 0.0
6	0.5, 0.5, 0.0
7	0.0, 0.5, 0.0
8	0.0, 0.0, 0.5
9	0.5, 0.0, 0.5
10	0.0, 0.5, 0.5
11	1/4, 1/4, 0.0
12	0.5, 1/4, 1/4
13	0.0, 1/4, 1/4
14	1/4, 0.0, 1/4

To obtain a Tet14 by degenerating a Hex27, the following are corresponding nodes:

Tet14	Hex27
1	1
2	5
3	7
4	8
5	13
6	18
7	15
8	16
9	20
10	19
11	25
12	23
13	27
14	24

For more information, see Tet14 (p. 382).

Specific Data - Tet15

Element name = Tet15

Number of nodes = 15

Order = Quadratic

Degenerate element name = <none>

Table 15-44 Tet15 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	5	2
2	2	6	3
3	3	7	1
4	1	8	4
5	2	9	4
6	3	10	4

Table 15-45 Tet15 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 0.0
4	0.0, 0.0, 1.0
5	0.5, 0.0, 0.0
6	0.5, 0.5, 0.0
7	0.0, 0.5, 0.0
8	0.0, 0.0, 0.5
9	0.5, 0.0, 0.5
10	0.0, 0.5, 0.5
11	1/4, 1/4, 1/4
12	1/4, 1/4, 0.0
13	0.5, 1/4, 1/4
14	0.0, 1/4, 1/4
15	1/4, 0.0, 1/4

To obtain a Tet15 by degenerating a Hex27, the following are corresponding nodes:

Tet15	Hex27
1	1
2	5
3	7
4	8
5	13
6	18
7	15
8	16
9	20
10	19
11	21
12	25
13	23
14	27
15	24

For more information, see **Tet15** (p. 385).

Specific Data - Tet16

Element name = Tet16

Number of nodes = 16

Order = Cubic

Degenerate element name = <none>

Table 15-46 Tet16 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	5	6	2
2	2	7	8	3
3	3	9	10	1
4	1	11	14	4
5	2	12	15	4
6	3	13	16	4

Table 15-47 Tet16 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0
2	1.0, 0.0, 0.0
3	0.0, 1.0, 0.0
4	0.0, 0.0, 1.0
5	1 / 3, 0.0, 0.0
6	2 / 3, 0.0, 0.0
7	2 / 3,1 / 3, 0.0
8	1 / 3, 2 / 3, 0.0
9	0.0, 2 / 3, 0.0
10	0.0, 1 / 3, 0.0
11	0.0, 0.0, 1 / 3
12	2 / 3, 0.0, 1 / 3
13	0.0, 1 / 3, 2 / 3
14	0.0, 0.0, 2 / 3
15	1 / 3, 0.0, 2 / 3
16	0.0, 1 / 3, 2 / 3

To obtain a Tet16 by degenerating a Hex32, the following are corresponding nodes:

Tet16	Hex32
1	1
2	5
3	7
4	8
5	17
6	21
7	27
8	28
9	23
10	19
11	20
12	32

Tet16	Hex32
13	29
14	24
15	31
16	30

For more information, see Tet16 (p. 388).

Specific Data - Tet40

Element name = Tet40

Number of nodes = 40

Order = Cubic

Degenerate element name = <none>

Table 15-48 Tet40 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	5	6	2
2	2	7	8	3
3	3	9	10	1
4	1	11	14	4
5	2	12	15	4
6	3	13	16	4

Table 15-49 Tet40 Node Parametric Coordinates

Node Number	C(3)	Node Number	Xi/Eta/Zeta or L2/L3/L4
1	0.0, 0.0, 0.0	21	2/9, 0.0, 1/9
2	1.0, 0.0, 0.0	22	4/9, 0.0, 2/9
3	0.0, 1.0, 0.0	23	2/3, 2/9, 1/9
4	0.0, 0.0, 1.0	24	1/3, 4/9, 2/9
5	1/3, 0.0, 0.0	25	0.0, 1/9, 2/9
6	2/3, 0.0, 0.0	26	0.0, 2/9, 1/9
7	2/3, 1/3, 0.0	27	2/9,.074074,.037037
8	1/3, 2/3, 0.0	28	4/9, 148148,.074074
9	0.0, 2/3, 0.0	29	2/9, .296297,.148148

Table 15-49 Tet40 Node Parametric Coordinates

Node Number	C(3)	Node Number	Xi/Eta/Zeta or L2/L3/L4
10	0.0, 1/3,0.0	30	1/9,.148148, .074074
11	0.0, 0.0, 1/3	31	1/9, 0.0, 2/9
12	2/3, 0.0, 1/3	32	2/9, 0.0, 4/9
13	0.0, 2/3, 1/3	33	2/3, 1/9, 2/9
14	0.0, 0.0, 2/3	34	1/3, 2/9, 4/9
15	1/3, 0.0, 2/3	35	0.0, 2/9, 4/9
16	0.0, 1/3, 2/3	36	0.0, 4/9, 2/9
17	2/9, 1/9, 0.0	37	2/9,.037037,.074074
18	4/9, 2/9, 0.0	38	4/9,.074074,.148148
19	2/9, 4/9, 0.0	39	2/9,.148148,.296297
20	1/9, 2/9, 0.0	40	1/9,.074074,.148148

To obtain a Tet40 by degenerating a Hex64, the following are corresponding nodes:

Tet40	Hex64
1	1
2	5
3	7
4	8
5	17
6	21
7	27
8	28
9	23
10	19
11	20
12	32
13	29
14	24
15	31
16	30
17	39

Tet40	Hex64
18	51
19	52
20	40
21	44
22	56
23	62
24	63
25	42
26	41
27	46
28	58
29	59
30	47
31	43
32	55
33	61
34	64
35	54
36	53
37	45
38	57
39	60
40	48

For more information, see Tet40 (p. 390).

15.6 Wedge Element Topology

MSC.Patran contains eight different wedge element topologies: Wedge6, Wedge7, Wedge15, Wedge16, Wedge20, Wedge21, Wedge24 and Wedge52.

General Data

Shape = Wedge

Element dimensionality= 3

Number of corner nodes = 6

Number of edges = 9

Number of faces = 5

Number of face edges = 4,3

General Shape

For Wedge elements, a combination of area and rectangular coordinates [L1/L2/L3/Zeta] are commonly used. Zeta values vary from -1 to 1 as in a Hex element. The area coordinates L1/L2/L3 represent the weighting with respect to the 3 edges along the Zeta direction:

edge number 8 (node 1-->4)

edge number 7 (node 2-->5)

edge number 9 (node 3-->6)

See **Area** (p. 318) coordinate system for more information.

Wedge elements can be obtained by degenerating Hex elements.

1. Hex corner node 2 collapses onto 1, and 6 collapses onto 5.
2. Wedge corner nodes 1:6 match 1/3/4/5/7/8 for the Hex.

Table 15-50 Wedge Face Numbering

Face ID	Sense	Edge 1	Edge 2	Edge 3	Edge 4
1	1	1	2	3	*
2	-1	4	5	6	*
3	-1	1	8	4	7
4	-1	2	9	5	8
5	-1	3	7	6	9

Specific Data - Wedge6

Element name = Wedge6

Number of nodes = 6

Order = linear

Degenerate element name = Tet4

Table 15-51 Wedge6 Edge Numbering

Edge Number	Node 1	Node 2
1	2	1
2	1	3
3	3	2
4	5	4
5	4	6
6	6	5
7	2	5
8	1	4
9	3	6

Table 15-52 Wedge6 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0
6	0.0, 1.0, 1.0

To obtain a Wedge6 by degenerating a Hex8, the following are corresponding nodes:

Wedge6	Hex8
1	1
2	3
3	4
4	5
5	7
6	8

For more information, see Wedge 6 (p. 378).

Specific Data - Wedge7

Element name = Wedge7

Number of nodes = 7

Order = linear

Degenerate element name = Tet5

Table 15-53 Wedge7 Edge Numbering

Edge Number	Node 1	Node 2
1	2	1
2	1	3
3	3	2
4	5	4
5	4	6
6	6	5
7	2	5
8	1	4
9	3	6

Table 15-54 Wedge7 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0
6	0.0, 1.0, 1.0
7	1/3, 1/3, 0.0

To obtain a Wedge7 by degenerating a Hex9, the following are corresponding nodes:

Wedge7	Hex9
1	1
2	3
3	4
4	5
5	7
6	8
7	9

For more information, see **Wedge7** (p. 379).

Specific Data - Wedge15

Element name = Wedge15

Number of nodes = 15

Order = quadratic

Degenerate element name = Tet10

Table 15-55 Wedge15 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	2	7	1
2	1	9	3
3	3	8	2
4	5	13	4
5	4	15	6
6	6	14	5
7	2	11	5
8	1	10	4
9	3	12	6

Table 15-56 Wedge15 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0

Table 15-56 Wedge15 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0
6	0.0, 1.0, 1.0
7	0.5, 0.0, -1.0
8	0.5, 0.5, -1.0
9	0.0, 0.5, -1.0
10	0.0, 0.0, 0.0
11	1.0, 0.0, 0.0
12	0.0, 1.0, 0.0
13	0.5, 0.0, 1.0
14	0.5, 0.5, 1.0
15	0.0, 0.5, 1.0

To obtain a Wedge15 by degenerating a Hex20, the following are corresponding nodes:

Wedge15	Hex20
1	1
2	3
3	4
4	5
5	7
6	8
7	10
8	11
9	12
10	13
11	15
12	16
13	18
14	19
15	20

For more information, see Wedge15 (p. 380).

Specific Data - Wedge16

Element name = Wedge16

Number of nodes = 16

Order = quadratic

Degenerate element name = Tet11

Table 15-57 Wedge16 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	2	7	1
2	1	9	3
3	3	8	2
4	5	13	4
5	4	15	6
6	6	14	5
7	2	11	5
8	1	10	4
9	3	12	6

Table 15-58 Wedge16 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0
6	0.0, 1.0, 1.0
7	0.5, 0.0, -1.0
8	0.5, 0.5, -1.0
9	0.0, 0.5, -1.0
10	0.0, 0.0, 0.0
11	1.0, 0.0, 0.0

Table 15-58 Wedge16 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
12	0.0, 1.0, 0.0
13	0.5, 0.0, 1.0
14	0.5, 0.5, 1.0
15	0.0, 0.5, 1.0
16	1 / 3, 1 / 3, 0.0

To obtain a Wedge16 by degenerating a Hex21, the following are corresponding nodes:

Wedge16	Hex21
1	1
2	3
3	4
4	5
5	7
6	8
7	10
8	11
9	12
10	13
11	15
12	16
13	18
14	19
15	20
16	21

For more information, see Wedge16 (p. 381).

Specific Data - Wedge20

Element name = Wedge20

Number of nodes = 20

Order = quadratic

Degenerate element name = Tet14

Table 15-59 Wedge20 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	2	7	1
2	1	9	3
3	3	8	2
4	5	13	4
5	4	15	6
6	6	14	5
7	2	11	5
8	1	10	4
9	3	12	6

Table 15-60 Wedge20 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0
6	0.0, 1.0, 1.0
7	0.5, 0.0, -1.0
8	0.5, 0.5, -1.0
9	0.0, 0.5, -1.0
10	0.0, 0.0, 0.0
11	1.0, 0.0, 0.0
12	0.0, 1.0, 0.0
13	0.5, 0.0, 1.0
14	0.5, 0.5, 1.0
15	0.0, 0.5, 1.0
16	1/3, 1/3, -1.0

Table 15-60 Wedge20 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
17	1/3, 1/3, 1.0
18	0.5, 0.5, 0.0
19	0.0, 0.5, 0.0
20	0.5, 0.0, 0.0

To obtain a Wedge20 by degenerating a Hex26, the following are corresponding nodes:

Wedge20	Hex26
1	1
2	3
3	4
4	5
5	7
6	8
7	10
8	11
9	12
10	13
11	15
12	16
13	18
14	19
15	20
16	21
17	22
18	26
19	23
20	24

For more information, see **Wedge20** (p. 383).

Specific Data - Wedge21

Element name = Wedge21

Number of nodes = 21

Order = quadratic

Degenerate element name = Tet15

Table 15-61 Wedge21 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	2	7	1
2	1	9	3
3	3	8	2
4	5	13	4
5	4	15	6
6	6	14	5
7	2	11	5
8	1	10	4
9	3	12	6

Table 15-62 Wedge21 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0
6	0.0, 1.0, 1.0
7	0.5, 0.0, -1.0
8	0.5, 0.5, -1.0
9	0.0, 0.5, -1.0
10	0.0, 0.0, 0.0
11	1.0, 0.0, 0.0
12	0.0, 1.0, 0.0

Table 15-62 Wedge21 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
13	0.5, 0.0, 1.0
14	0.5, 0.5, 1.0
15	0.0, 0.5, 1.0
16	1/3, 1/3, 0.0
17	1/3, 1/3, -1.0
18	1/3, 1/3, 1.0
19	0.5, 0.5, 0.0
20	0.0, 0.5, 0.0
21	0.5, 0.0, 0.0

To obtain a Wedge21 by degenerating a Hex27, the following are corresponding nodes:

Wedge21	Hex27
1	1
2	3
3	4
4	5
5	7
6	8
7	10
8	11
9	12
10	13
11	15
12	16
13	18
14	19
15	20
16	21
17	22

Wedge21	Hex27
18	23
19	27
20	24
21	25

For more information, see Wedge21 (p. 386).

Specific Data - Wedge24

Element name = Wedge24

Number of nodes = 24

Order = Cubic

Degenerate element name = Tet16

Table 15-63 Wedge24 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	2	8	7	1
2	1	12	11	3
3	3	10	9	2
4	5	20	19	4
5	4	24	23	6
6	6	22	21	5
7	2	14	17	5
8	1	13	16	4
9	3	15	18	6

Table 15-64 Wedge24 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0
2	1.0, 0.0, -1.0
3	0.0, 1.0, -1.0
4	0.0, 0.0, 1.0
5	1.0, 0.0, 1.0

Table 15-64 Wedge24 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta
6	0.0, 1.0, 1.0
7	1/3, 0.0, -1.0
8	2/3, 0.0, -1.0
9	2/3, 1/3, -1.0
10	1/3, 2/3, -1.0
11	0.0, 2/3, -1.0
12	0.0, 1/3, -1.0
13	0.0, 0.0, -1/3
14	1.0, 0.0, -1/3
15	0.0, 1.0, -1/3
16	0.0, 0.0, 1/3
17	1.0, 0.0, 1/3
18	0.0, 1.0, 1/3
19	1/3, 0.0, 1.0
20	2/3, 0.0, 1.0
21	2/3, 1/3, 1.0
22	1/3, 2/3, 1.0
23	0.0, 2/3, 1.0
24	0.0, 1/3, 1.0

To obtain a Wedge24 by degenerating a Hex32, the following are corresponding nodes:

Wedge24	Hex32
1	1
2	3
3	4
4	5
5	7
6	8
7	11

Wedge24	Hex32
8	12
9	13
10	14
11	15
12	16
13	17
14	19
15	20
16	21
17	23
18	24
19	27
20	28
21	29
22	30
23	31
24	32

For more information, see Wedge24 (p. 388).

Specific Data - Wedge52

Element name = Wedge52

Number of nodes = 52

Order = Cubic

Degenerate element name = Tet40

Table 15-65 Wedge52 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	2	8	7	1
2	1	12	11	3
3	3	10	9	2
4	5	20	19	4
5	4	24	23	6

Table 15-65 Wedge52 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
6	6	22	21	5
7	2	14	17	5
8	1	13	16	4
9	3	15	18	6

Table 15-66 Wedge52 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta	Node Number	Xi/Eta/Zeta or L2/L3/Zeta
1	0.0, 0.0, -1.0	27	2/9, 4/9, -1.0
2	1.0, 0.0, -1.0	28	1/9, 2/9, -1.0
3	0.0, 1.0, -1.0	29	1/3, 0.0, -1/3
4	0.0, 0.0, 1.0	30	2/3, 0.0, -1/3
5	1.0, 0.0, 1.0	31	2/3, 1/3, -1/3
6	0.0, 1.0, 1.0	32	1/3, 2/3, -1/3
7	1/3, 0.0, 1.0	33	0.0, 2/3, -1/3
8	2/3, 0.0, -1.0	34	0.0, 1/3, -1/3
9	1/3, 1/3, -1.0	35	2/9, 1/9, -1/3
10	1/3, 2/3, -1.0	36	4/9, 2/9, -1/3
11	0.0, 2/3, -1.0	37	2/9, 4/9, -1/3
12	0.0, 1/3, -1.0	38	1/9, 2/9, -1/3
13	0.0, 0.0, -1/3	39	1/3, 0.0, 1/3
14	1.0, 0.0, -1/3	40	2/3, 0.0, 1/3
15	0.0, 1.0, -1/3	41	2/3, 1/3, 1/3
16	0.0, 0.0, 1/3	42	1/3, 2/3, 1/3
17	1.0, 0.0, 1/3	43	0.0, 2/3, 1/3
18	0.0, 1.0, 1/3	44	0.0, 1/3, 1/3
19	1/3, 0.0, 1.0	45	2/9, 1/9, 1/3
20	2/3, 0.0, 1.0	46	4/9, 2/9, 1/3
21	2/3, 1/3, 1.0	47	2/9, 4/9, 1/3
22	1/3, 2/3, 1.0	48	1/9, 2/9, 1/3
23	0.0, 2/3, 1.0	49	2/9, 1/9, 1.0
24	0.0, 1/3, 1.0	50	4/9, 2/9, 1.0

Table 15-66 Wedge52 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta or L2/L3/Zeta	Node Number	Xi/Eta/Zeta or L2/L3/Zeta
25	2 / 9, 1 / 9, -1.0	51	2 / 9, 4 / 9, 1.0
26	4 / 9, 2 / 9, -1.0	52	1 / 9, 2 / 9, 1.0

To obtain a Wedge52 by degenerating a Hex64, the following are corresponding nodes:

Wedge52	Hex64
1	1
2	3
3	4
4	5
5	7
6	8
7	11
8	12
9	13
10	14
11	15
12	16
13	17
14	19
15	20
16	21
17	23
18	24
19	27
20	28
21	29
22	30
23	31
24	32
25	34

Wedge52	Hex64
26	35
27	36
28	33
29	39
30	40
31	41
32	42
33	43
34	44
35	46
36	47
37	48
38	45
39	51
40	52
41	53
42	54
43	55
44	56
45	58
46	59
47	60
48	57
49	62
50	63
51	64
52	61

For more information, see **Wedge 52** (p. 391).

15.7 Hex Element Topology

MSC.Patran contains eight different hex element topologies: Hex8, Hex9, Hex20, Hex21, Hex26, Hex27, Hex32, Hex64.

General Data

Shape = Hex

Element dimensionality= 3

Number of corner nodes = 8

Number of edges = 12

Number of faces = 6

Number of face edges = 4

General Shape

TheHex parametric coordinates (Rectangular) are:

1. X axis for the Hex element is from node 1-->2.
2. Y axis for the Hex element is from node 1-->4.
3. Z axis for the Hex element is from node 1-->5.

Table 15-67 Hex Face Numbering

Face ID	Sense	Edge 1	Edge 2	Edge 3	Edge 4
1	1	1	2	3	4
2	-1	5	6	7	8
3	-1	1	10	5	9
4	-1	2	11	6	10
5	-1	3	12	7	11
6	-1	4	9	8	12

Specific Data - Hex8

Element name = Hex8

Number of nodes = 8

Order = linear

Degenerate element name = Wedge6

Table 15-68 Hex8 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	6
3	6	5
4	5	1
5	4	3
6	3	7
7	7	8
8	8	4
9	1	4
10	2	3
11	6	7
12	5	8

Table 15-69 Hex8 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0
2	1.0, -1.0, -1.0
3	1.0, 1.0, -1.0
4	-1.0, 1.0, -1.0
5	-1.0, -1.0, 1.0
6	1.0, -1.0, 1.0
7	1.0, 1.0, 1.0
8	-1.0, 1.0, 1.0

For more information, see **Hex8** (p. 378).

Specific Data - Hex9

Element name = Hex9

Number of nodes = 9

Order = linear

Degenerate element name = Wedge7

Table 15-70 Hex9 Edge Numbering

Edge Number	Node 1	Node 2
1	1	2
2	2	6
3	6	5
4	5	1
5	4	3
6	3	7
7	7	8
8	8	4
9	1	4
10	2	3
11	6	7
12	5	8

Table 15-71 Hex9 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0
2	1.0, -1.0, -1.0
3	1.0, 1.0, -1.0
4	-1.0, 1.0, -1.0
5	-1.0, -1.0, 1.0
6	1.0, -1.0, 1.0
7	1.0, 1.0, 1.0
8	-1.0, 1.0, 1.0
9	0.0, 0.0, 0.0

For more information, see **Hex9** (p. 379).

Specific Data - Hex20

Element name = Hex20

Number of nodes = 20

Order = quadratic

Degenerate element name = Wedge15

Table 15-72 Hex20 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	9	2
2	2	14	6
3	6	17	5
4	5	13	1
5	4	11	3
6	3	15	7
7	7	19	8
8	8	16	4
9	1	12	4
10	2	10	3
11	6	18	7
12	5	20	8

Table 15-73 Hex20 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0
2	1.0, -1.0, -1.0
3	1.0, 1.0, -1.0
4	-1.0, 1.0, -1.0
5	-1.0, -1.0, 1.0
6	1.0, -1.0, 1.0
7	1.0, 1.0, 1.0
8	-1.0, 1.0, 1.0
9	0.0, -1.0, -1.0
10	1.0, 0.0, -1.0

Table 15-73 Hex20 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
11	0.0, 1.0, -1.0
12	-1.0, 0.0, -1.0
13	-1.0, -1.0, 0.0
14	1.0, -1.0, 0.0
15	1.0, 1.0, 0.0
16	-1.0, 1.0, 0.0
17	0.0, -1.0, 1.0
18	1.0, 0.0, 1.0
19	0.0, 1.0, 1.0
20	-1.0, 0.0, 1.0

For more information, see **Hex20** (p. 380).

Specific Data - Hex21

Element name = Hex21

Number of nodes = 21

Order = quadratic

Degenerate element name = Wedge16

Table 15-74 Hex21 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	9	2
2	2	14	6
3	6	17	5
4	5	13	1
5	4	11	3
6	3	15	7
7	7	19	8
8	8	16	4
9	1	12	4
10	2	10	3

Table 15-74 Hex21 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
11	6	18	7
12	5	20	8

Table 15-75 Hex21 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0
2	1.0, -1.0, -1.0
3	1.0, 1.0, -1.0
4	-1.0, 1.0, -1.0
5	-1.0, -1.0, 1.0
6	1.0, -1.0, 1.0
7	1.0, 1.0, 1.0
8	-1.0, 1.0, 1.0
9	0.0, -1.0, -1.0
10	1.0, 0.0, -1.0
11	0.0, 1.0, -1.0
12	-1.0, 0.0, -1.0
13	-1.0, -1.0, 0.0
14	1.0, -1.0, 0.0
15	1.0, 1.0, 0.0
16	-1.0, 1.0, 0.0
17	0.0, -1.0, 1.0
18	1.0, 0.0, 1.0
19	0.0, 1.0, 1.0
20	-1.0, 0.0, 1.0
21	0.0, 0.0, 0.0

For more information, see **Hex21** (p. 381).

Specific Data - Hex26

Element name = Hex26

Number of nodes = 26

Order = quadratic

Degenerate element name = Wedge20

Table 15-76 Hex26 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	9	2
2	2	14	6
3	6	17	5
4	5	13	1
5	4	11	3
6	3	15	7
7	7	19	8
8	8	16	4
9	1	12	4
10	2	10	3
11	6	18	7
12	5	20	8

Table 15-77 Hex26 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0
2	1.0, -1.0, -1.0
3	1.0, 1.0, -1.0
4	-1.0, 1.0, -1.0
5	-1.0, 1.0, 1.0
6	1.0, -1.0, 1.0
7	1.0, 1.0, 1.0
8	-1.0, 1.0, 1.0

Table 15-77 Hex26 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
9	0.0, -1.0, -1.0
10	1.0, 0.0, -1.0
11	0.0, 1.0, -1.0
12	-1.0, 0.0, -1.0
13	-1.0, -1.0, 0.0
14	1.0, -1.0, 0.0
15	1.0, 1.0, 0.0
16	-1.0, 1.0, 0.0
17	0.0, -1.0, 1.0
18	1.0, 0.0, 1.0
19	0.0, 1.0, 1.0
20	-1.0, 0.0, 1.0
21	0.0, 0.0, -1.0
22	0.0, 0.0, -1.0
23	-1.0, 0.0, 0.0
24	1.0, 0.0, 0.0
25	0.0, -1.0, 0.0
26	0.0, 1.0, 0.0

For more information, see **Hex26** (p. 384).

Specific Data - Hex27

Element name = Hex27

Number of nodes = 27

Order = quadratic

Degenerate element name = Wedge21

Table 15-78 Hex27 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
1	1	9	2
2	2	14	6
3	6	17	5

Table 15-78 Hex27 Edge Numbering

Edge Number	Node 1	Node 2	Node 3
4	5	13	1
5	4	11	3
6	3	15	7
7	7	19	8
8	8	16	4
9	1	12	4
10	2	10	3
11	6	18	7
12	5	20	8

Table 15-79 Hex27 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0
2	1.0, -1.0, -1.0
3	1.0, 1.0, -1.0
4	-1.0, 1.0, -1.0
5	-1.0, -1.0, 1.0
6	1.0, -1.0, 1.0
7	1.0, 1.0, 1.0
8	-1.0, 1.0, 1.0
9	0.0, -1.0, -1.0
10	1.0, 0.0, -1.0
11	0.0, 1.0, -1.0
12	-1.0, 0.0, -1.0
13	-1.0, -1.0, 0.0
14	1.0, -1.0, 0.0
15	1.0, 1.0, 0.0
16	-1.0, 1.0, 0.0
17	0.0, -1.0, 1.0

Table 15-79 Hex27 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta
18	1.0, 0.0, 1.0
19	0.0, 1.0, 1.0
20	-1.0, 0.0, 1.0
21	0.0, 0.0, 0.0
22	0.0, 0.0, -1.0
23	0.0, 0.0, 1.0
24	-1.0, 0.0, 0.0
25	1.0, 0.0, 0.0
26	0.0, -1.0, 0.0
27	0.0, 1.0, 0.0

For more information, see **Hex27** (p. 387).

Specific Data - Hex32

Element name = Hex32

Number of nodes = 32

Order = cubic

Degenerate element name = Wedge24

Table 15-80 Hex32 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	9	10	2
2	2	18	22	6
3	6	26	25	5
4	5	21	17	1
5	4	14	13	3
6	3	19	23	7
7	7	29	30	8
8	8	24	20	4
9	1	16	15	4
10	2	11	12	3

Table 15-80 Hex32 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
11	6	27	28	7
12	5	32	31	8

Table 15-81 Hex32 Node Parametric Coordinates

Node Number	Xi/Eta/Zeta	Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0	17	-1.0, -1.0, -1/3
2	1.0, -1.0, -1.0	18	1.0, -1.0, -1/3
3	1.0, 1.0, -1.0	19	1.0, 1.0, -1/3
4	-1.0, 1.0, -1.0	20	-1.0, 1.0, -1/3
5	-1.0, -1.0, 1.0	21	-1.0, -1.0, 1/3
6	1.0, -1.0, 1.0	22	1.0, -1.0, 1/3
7	1.0, 1.0, 1.0	23	1.0, 1.0, 1/3
8	-1.0, 1.0, 1.0	24	-1.0, 1.0, 1/3
9	-1/3, -1.0, -1.0	25	-1/3, -1.0, 1.0
10	1/3, -1.0, -1.0	26	1/3, -1.0, 1.0
11	1.0, -1/3, -1.0	27	1.0, -1/3, 1.0
12	1.0, 1/3, -1.0	28	1.0, 1/3, 1.0
13	1/3, 1.0, -1.0	29	1/3, 1.0, 1.0
14	-1/3, 1.0, -1.0	30	-1/3, 1.0, 1.0
15	-1.0, 1/3, -1.0	31	-1.0, 1/3, 1.0
16	-1.0, -1/3, -1.0	32	-1.0, -1/3, 1.0

For more information, see **Hex32** (p. 388).

Specific Data - Hex64

Element name = Hex64

Number of nodes = 64

Order = cubic

Degenerate element name = Wedge52

Table 15-82 Hex64 Edge Numbering

Edge Number	Node 1	Node 2	Node 3	Node 4
1	1	9	10	2
2	2	18	22	6
3	6	26	25	5
4	5	21	17	1
5	4	14	13	3
6	3	19	23	7
7	7	29	30	8
8	8	24	20	4
9	1	16	15	4
10	2	11	12	3
11	6	27	28	7
12	5	32	31	8

Table 15-83 Hex64 Node Parametric Coordinates

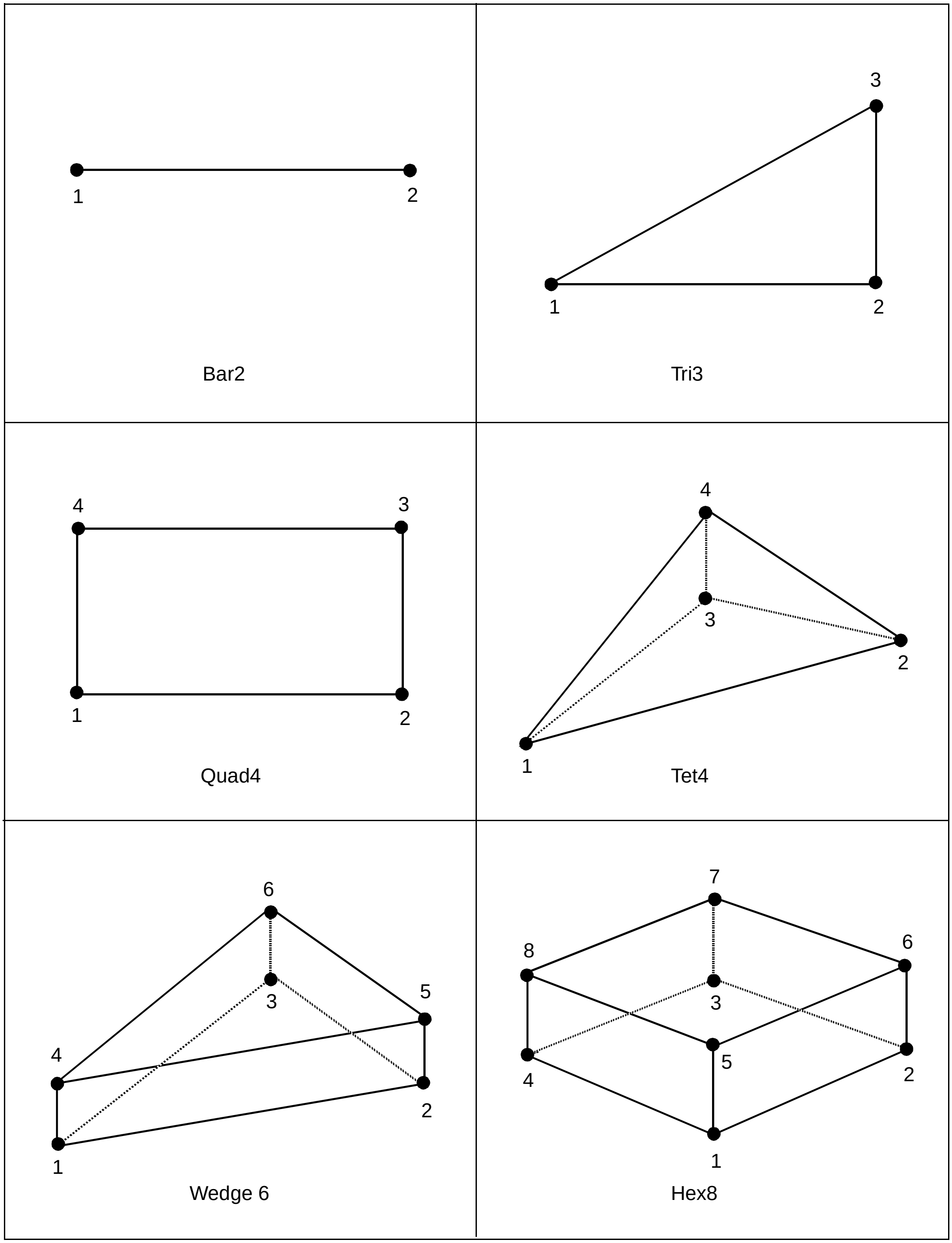
Node Number	Xi/Eta/Zeta	Node Number	Xi/Eta/Zeta	Node Number	Xi/Eta/Zeta
1	-1.0, -1.0, -1.0	23	1.0, 1.0, 1/3	45	-1/3, -1/3, -1/3
2	1.0, -1.0, -1.0	24	-1.0, 1.0, 1/3	46	1/3, -1/3, -1/3
3	1.0, 1.0, -1.0	25	-1/3, -1.0, 1.0	47	1/3, 1/3, -1/3
4	-1.0, 1.0, -1.0	26	1/3, -1.0, 1.0	48	-1/3, 1/3, -1/3
5	-1.0, -1.0, 1.0	27	1.0, -1/3, 1.0	49	-1/3, -1.0, 1/3
6	1.0, -1.0, 1.0	28	1.0, 1/3, 1.0	50	1/3, -1.0, 1/3
7	1.0, 1.0, 1.0	29	1/3, 1.0, 1.0	51	1.0, -1/3, 1/3
8	-1.0, 1.0, 1.0	30	-1/3, 1.0, 1.0	52	1.0, 1/3, 1/3
9	-1/3, -1.0, -1.0	31	-1.0, 1/3, 1.0	53	1/3, 1.0, 1/3
10	1/3, -1.0, -1.0	32	-1.0, -1/3, -1.0	54	-1/3, 1.0, 1/3
11	1.0, -1/3, -1.0	33	-1/3, -1/3, -1.0	55	-1.0, 1/3, 1/3
12	1.0, 1/3, -1.0	34	1/3, -1/3, -1.0	56	-1.0, -1/3, 1/3
13	1/3, 1.0, -1.0	35	1/3, 1/3, -1.0	57	-1/3, -1/3, 1/3

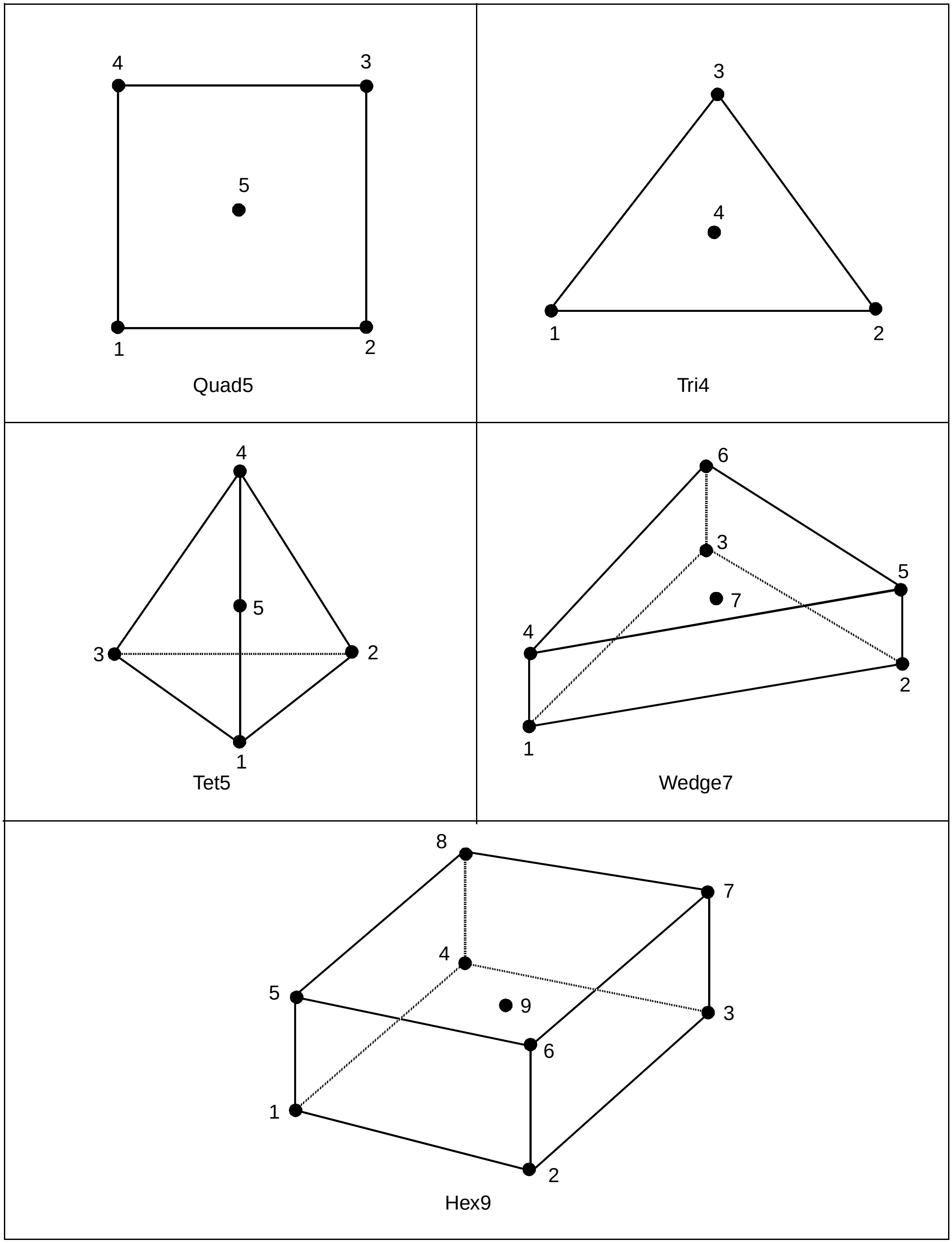
Table 15-83 Hex64 Node Parametric Coordinates

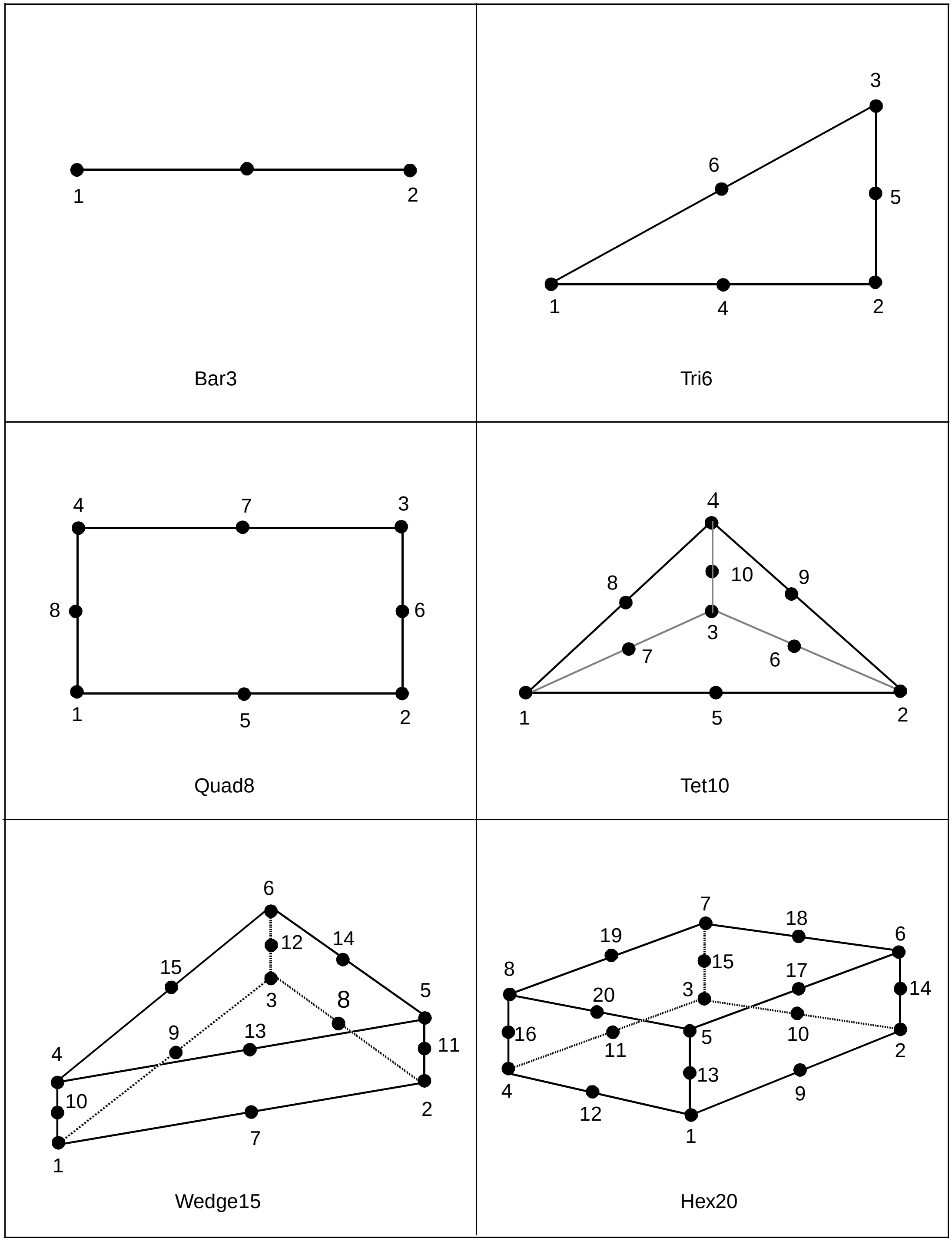
Node Number	Xi/Eta/Zeta	Node Number	Xi/Eta/Zeta	Node Number	Xi/Eta/Zeta
14	-1 / 3, 1.0, -1.0	36	-1 / 3, 1 / 3, -1.0	58	1 / 3, -1 / 3, 1 / 3
15	-1.0,1 / 3, -1.0	37	-1 / 3, -1.0, -1 / 3	59	1 / 3, 1 / 3, 1 / 3
16	-1.0, -1 / 3, -1.0	38	1 / 3, -1.0, -1 / 3	60	-1 / 3, 1 / 3, 1 / 3
17	-1.0, -1.0, -1 / 3	39	1.0, -1 / 3, -1 / 3	61	-1 / 3, -1 / 3, 1.0
18	1.0, -1.0, -1 / 3	40	1.0, 1 / 3, -1 / 3	62	1 / 3, -1 / 3, 1.0
19	1.0, 1.0, -1 / 3	41	1 / 3, 1.0, -1 / 3	63	1 / 3, 1 / 3, 1.0
20	-1.0, 1.0, -1 / 3	42	-1 / 3, 1.0, -1 / 3	64	-1 / 3, 1 / 3, 1.0
21	-1.0, -1.0, 1 / 3	43	-1.0, 1 / 3, -1 / 3		
22	1.0, -1.0, 1 / 3	44	-1.0, -1 / 3, -1 / 3		

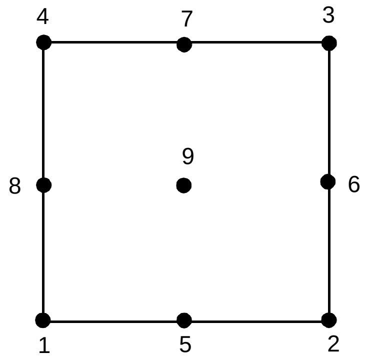
For more information, see **Hex64** (p. 392).

15.8 MSC.Patran's Element Library

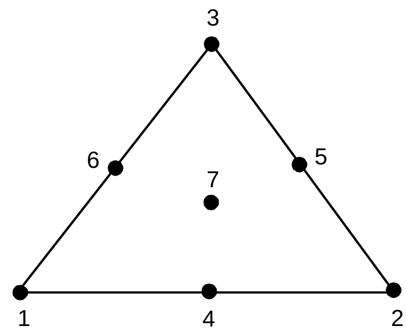



4
3
5
1
2
Quad5
3
4
1
2
Tri4
4
5
3
2
1
Tet5
6
3
5
7
4
2
1
Wedge7
8
7
4
5
9
3
6
1
2
Hex9

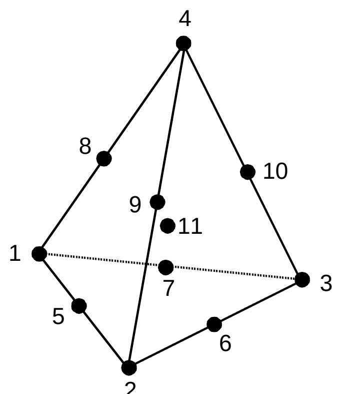

Bar3
Tri6
Quad8
Tet10
Wedge15
Hex20



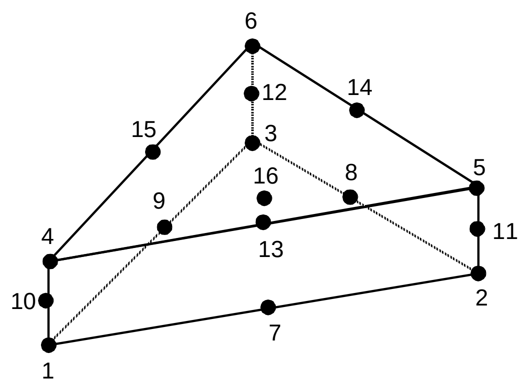
Quad9



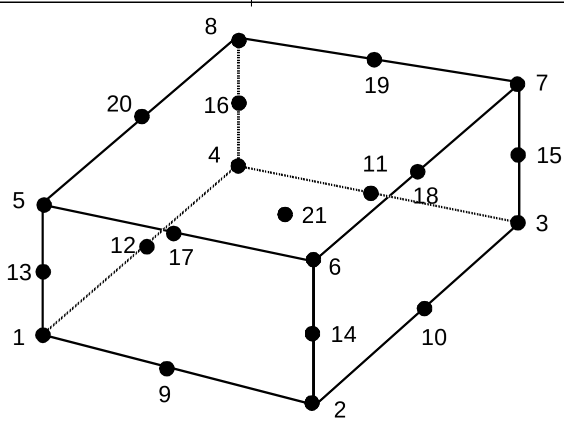
Tri7



Tet11

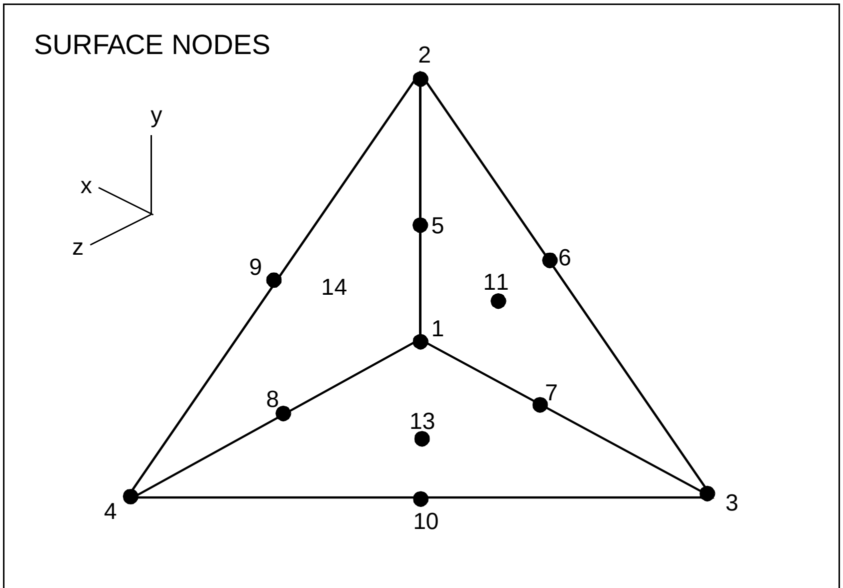


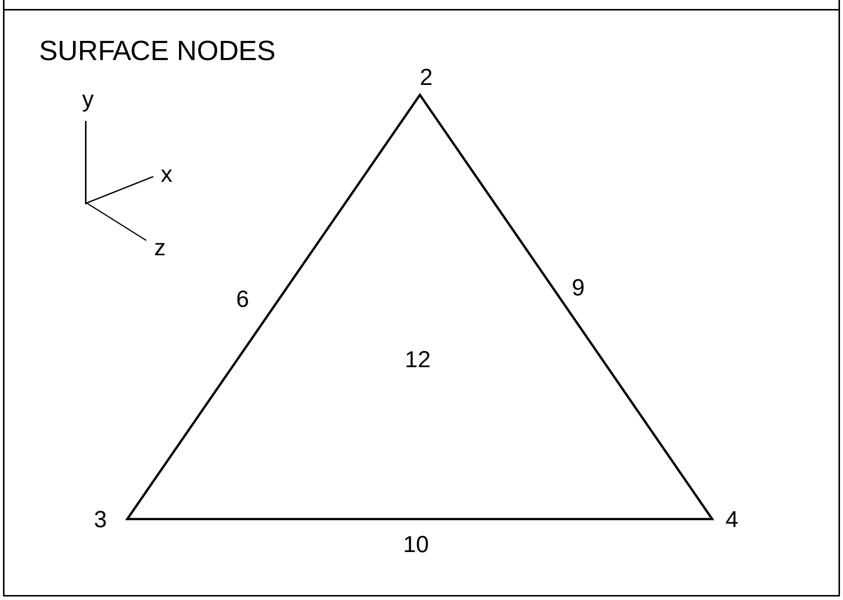
Wedge16



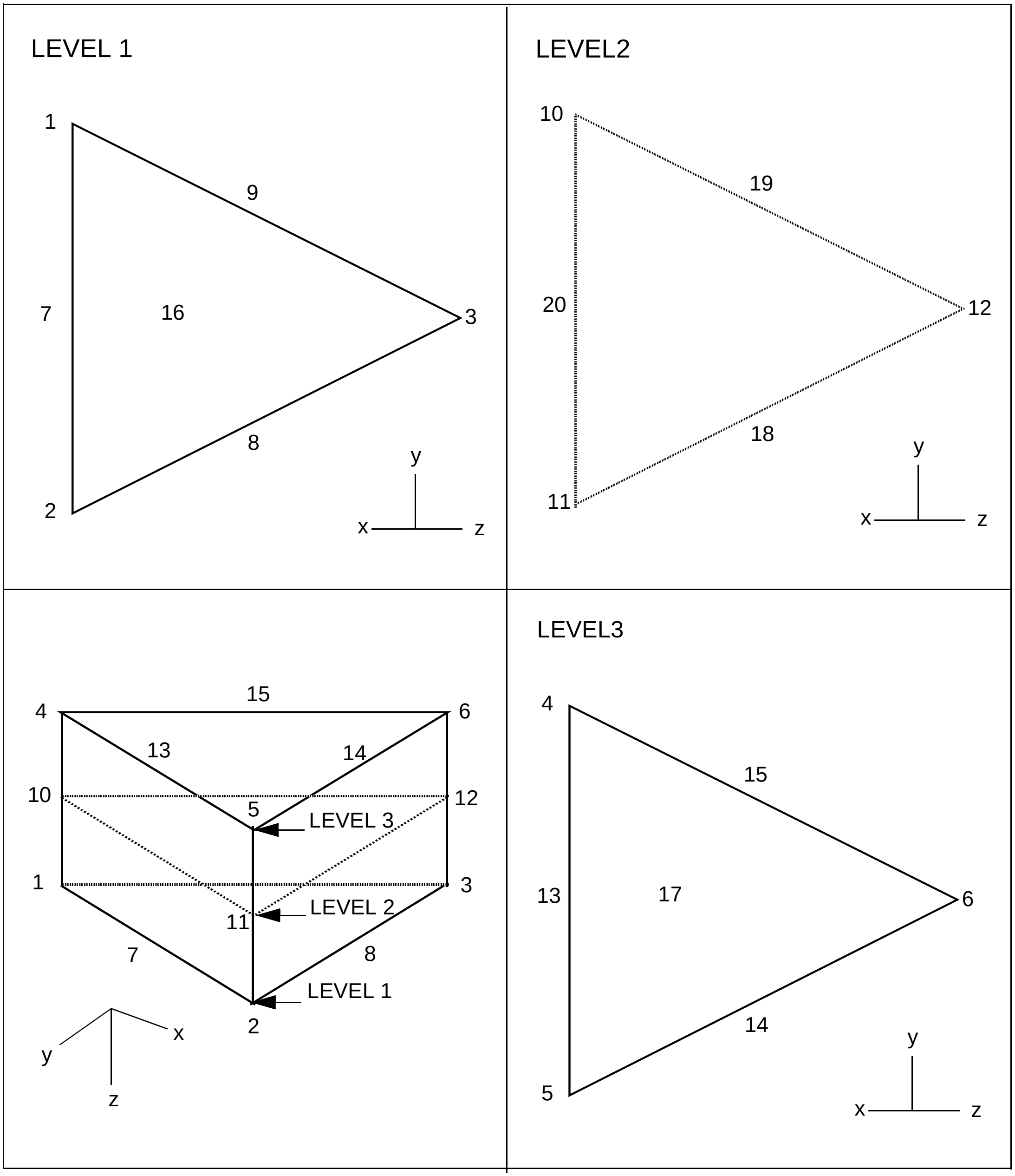
Hex21

Tet14

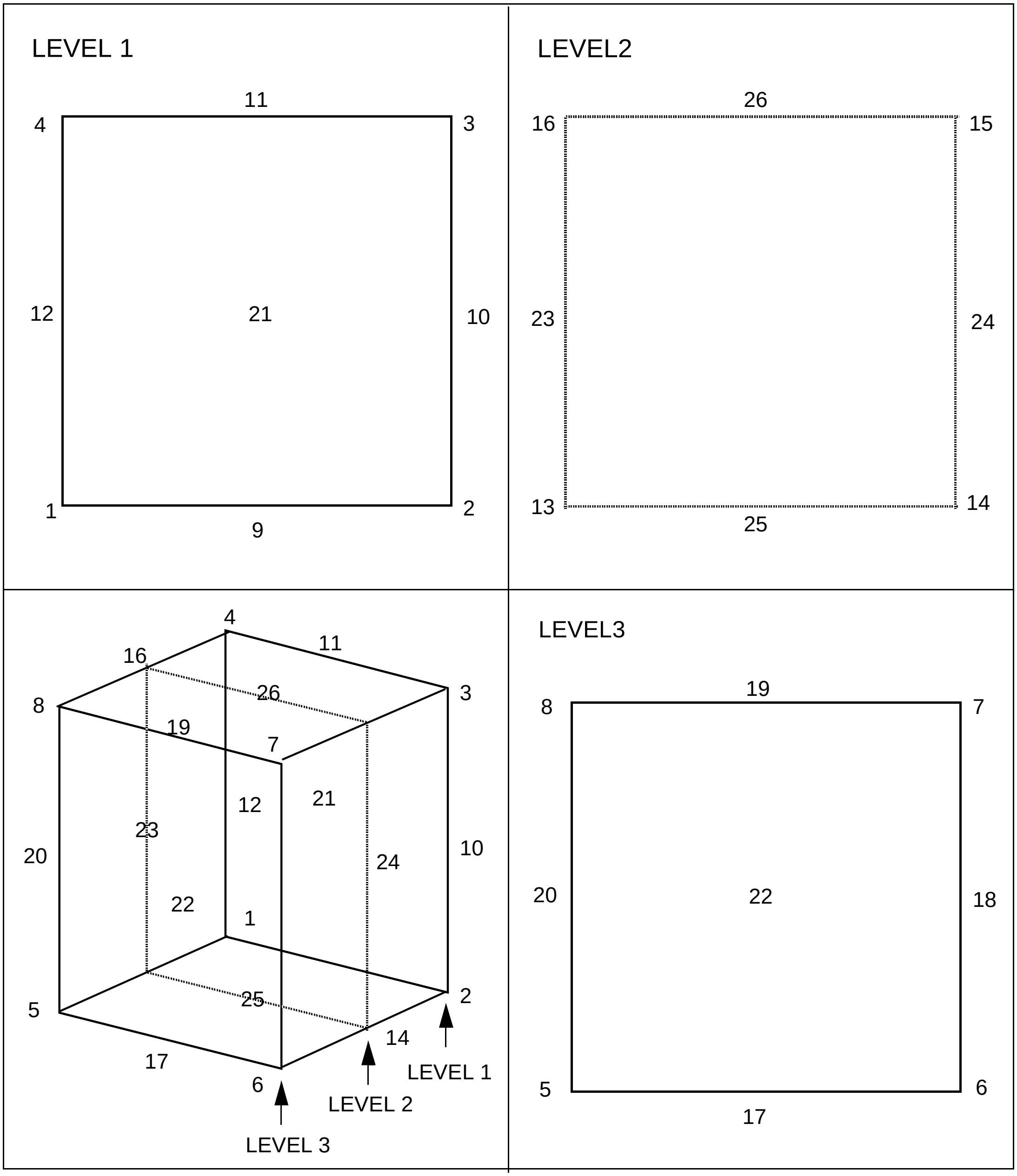

SURFACE NODES
y
x
z
2
5
6
9
14
11
1
8
7
13
4
10
3


SURFACE NODES
2
y
x
z
6
9
12
3
4
10

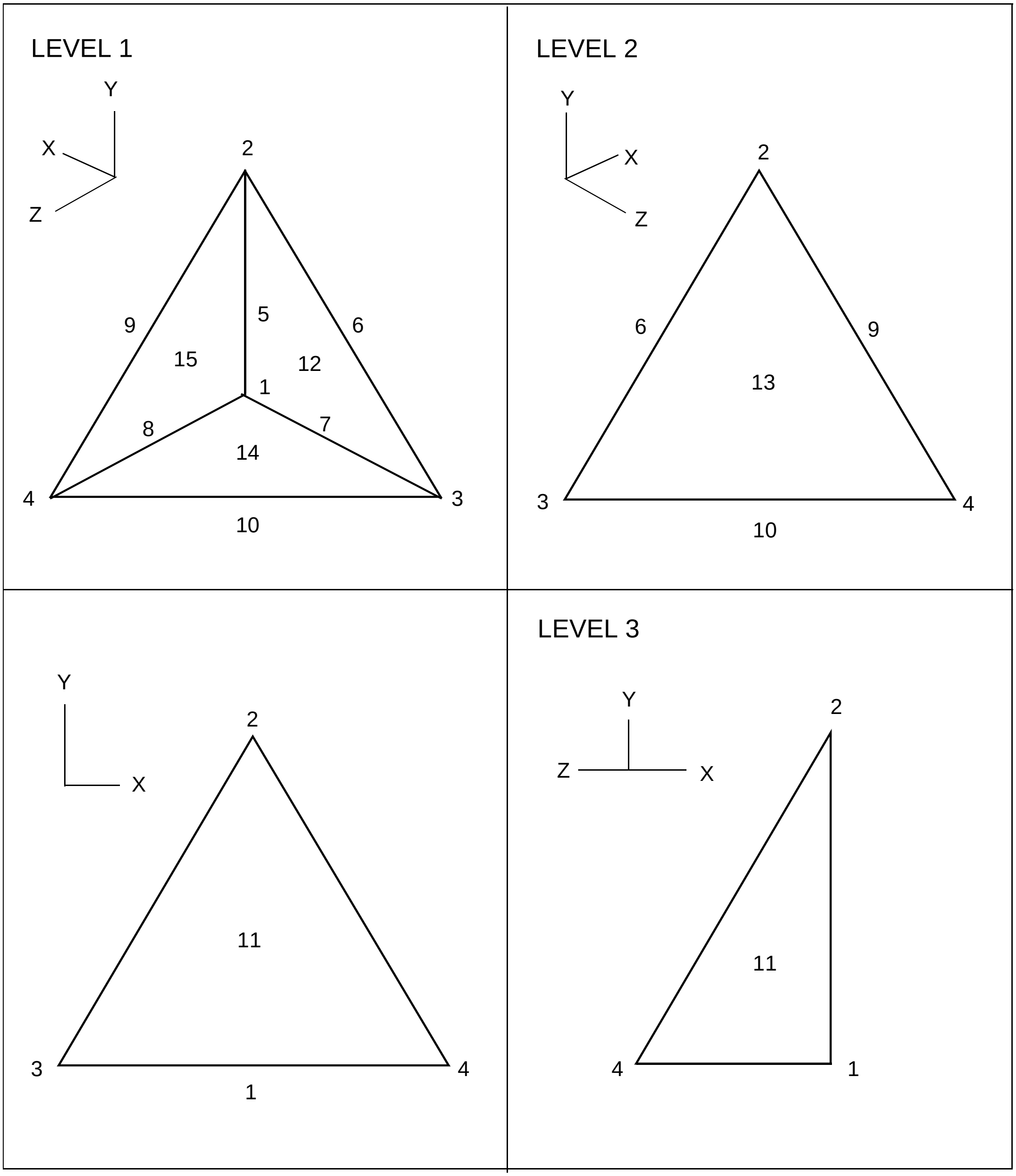
Wedge20


LEVEL 1
1
9
7
16
3
8
y
2
x
z
LEVEL2
10
19
20
12
18
y
11
x
z
15
4
6
13
14
10
12
5
LEVEL 3
1
3
LEVEL 2
11
7
8
LEVEL 1
2
x
y
z
LEVEL3
4
15
13
17
6
14
y
5
x
z

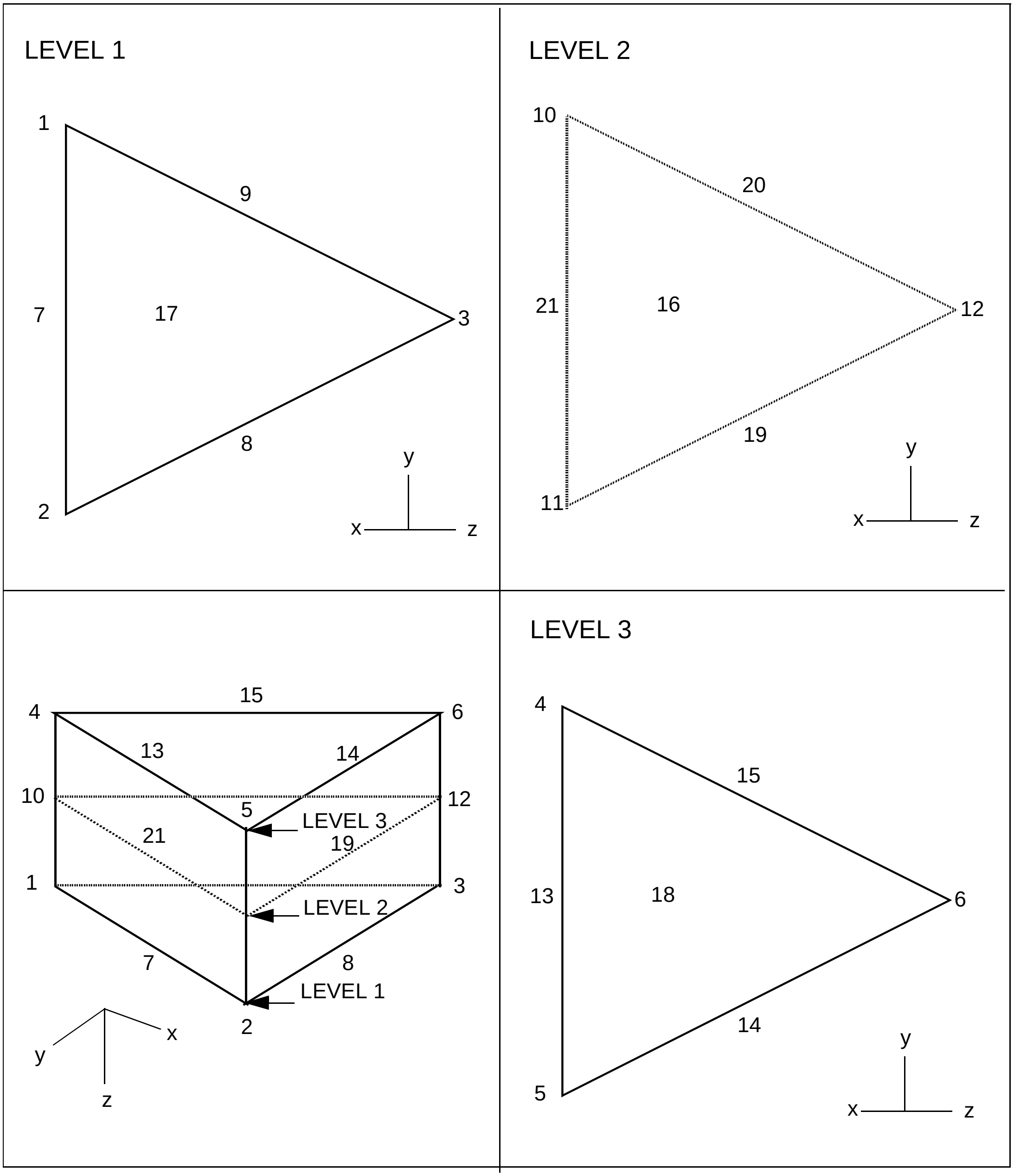
Hex26


LEVEL 1
11
4
3
12
21
10
1
2
9
LEVEL2
26
16
15
23
24
13
14
25
4
11
16
26
8
3
19
7
12
21
23
20
10
24
22
1
5
25
2
14
17
LEVEL 1
6
LEVEL 2
LEVEL 3
LEVEL3
19
8
7
20
22
18
5
6
17

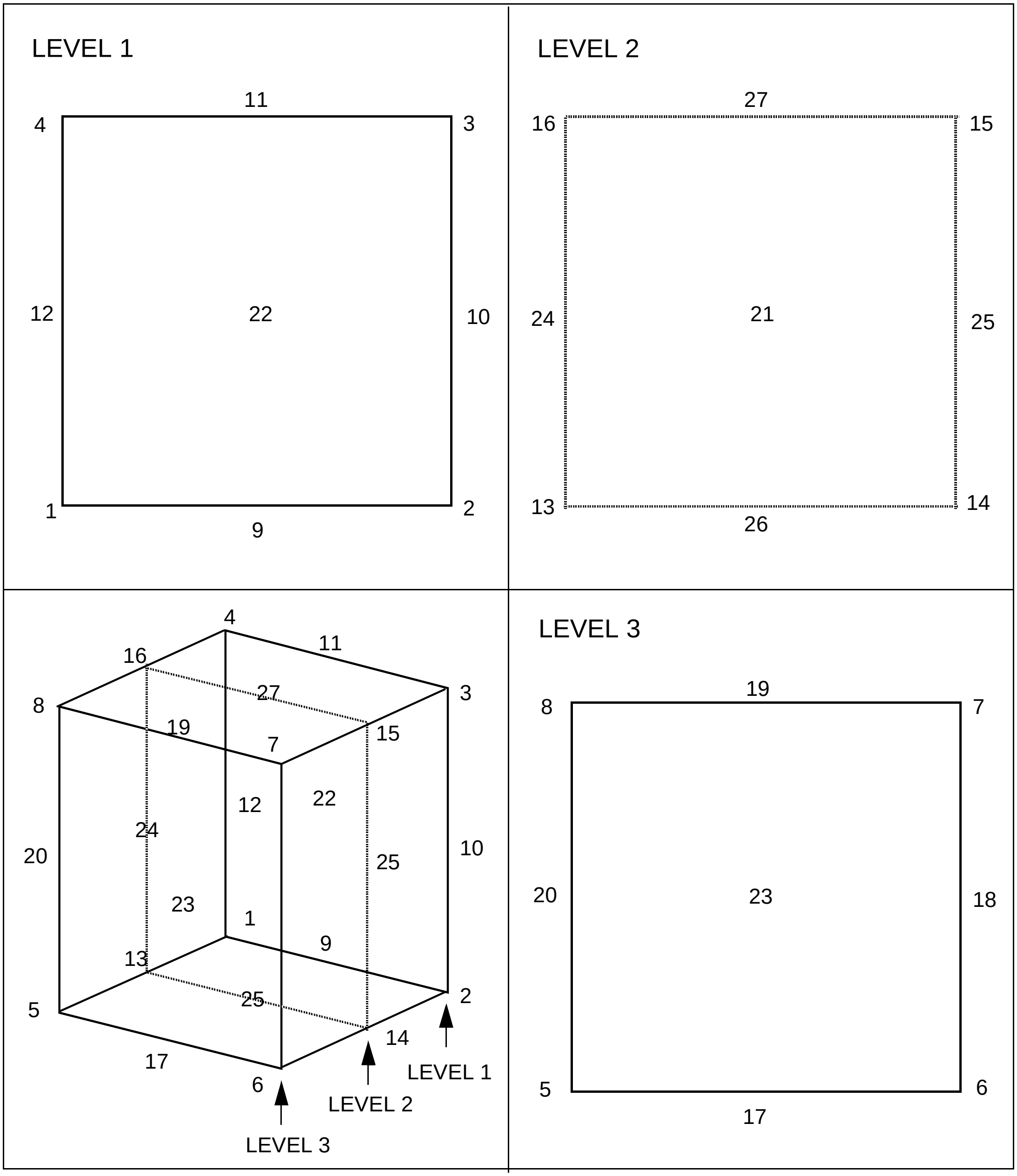
Tet15

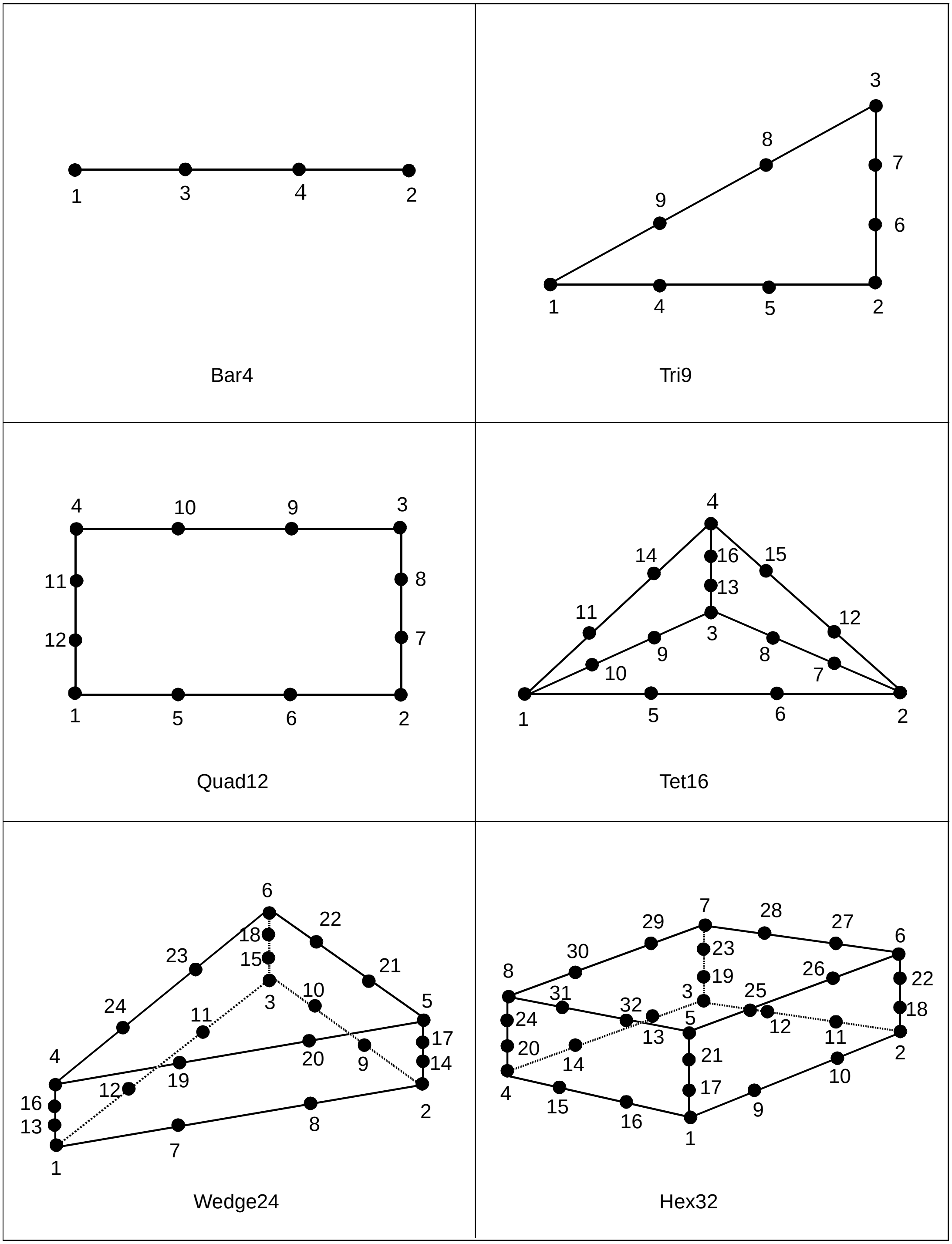

LEVEL 1
Y
X
Z
2
9
5
6
15
12
1
8
7
14
4
3
10
LEVEL 2
Y
X
Z
2
6
9
13
3
4
10
Y
X
2
11
3
4
1
LEVEL 3
Y
Z
X
2
11
4
1

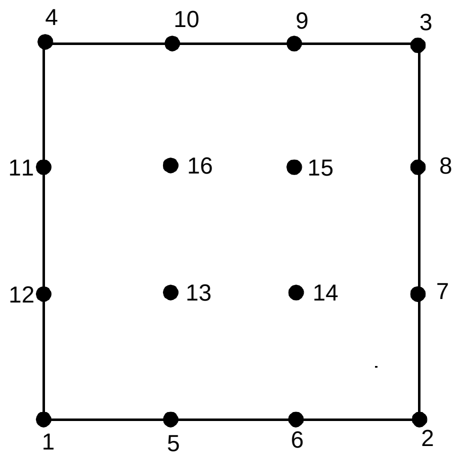
Wedge21


LEVEL 1
1
9
7
17
3
8
2
y
x
z
LEVEL 2
10
20
21
16
12
19
11
y
x
z
4
15
6
13
14
10
5
12
LEVEL 3
21
19
1
3
LEVEL 2
7
8
LEVEL 1
2
x
y
z
LEVEL 3
4
15
13
18
6
14
5
y
x
z

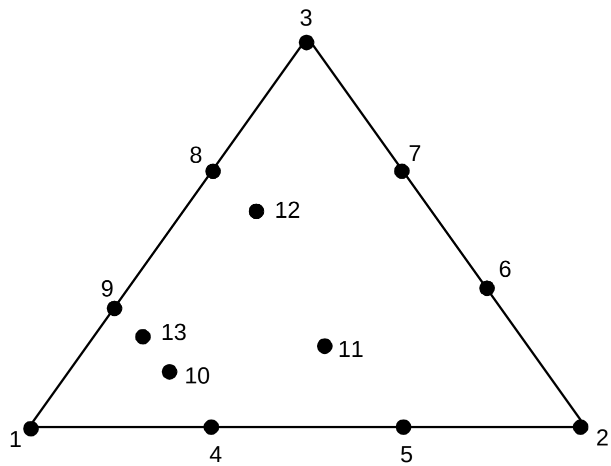
Hex27


LEVEL 1
11
4
3
12
22
10
1
2
9
LEVEL 2
27
16
15
24
21
25
13
14
26
4
16
11
8
27
3
19
7
15
12
22
24
20
10
25
23
1
9
13
5
25
2
14
17
LEVEL 1
6
LEVEL 2
LEVEL 3
LEVEL 3
19
8
7
20
23
18
5
6
17


Bar4
Tri9
Quad12
Tet16
Wedge24
Hex32

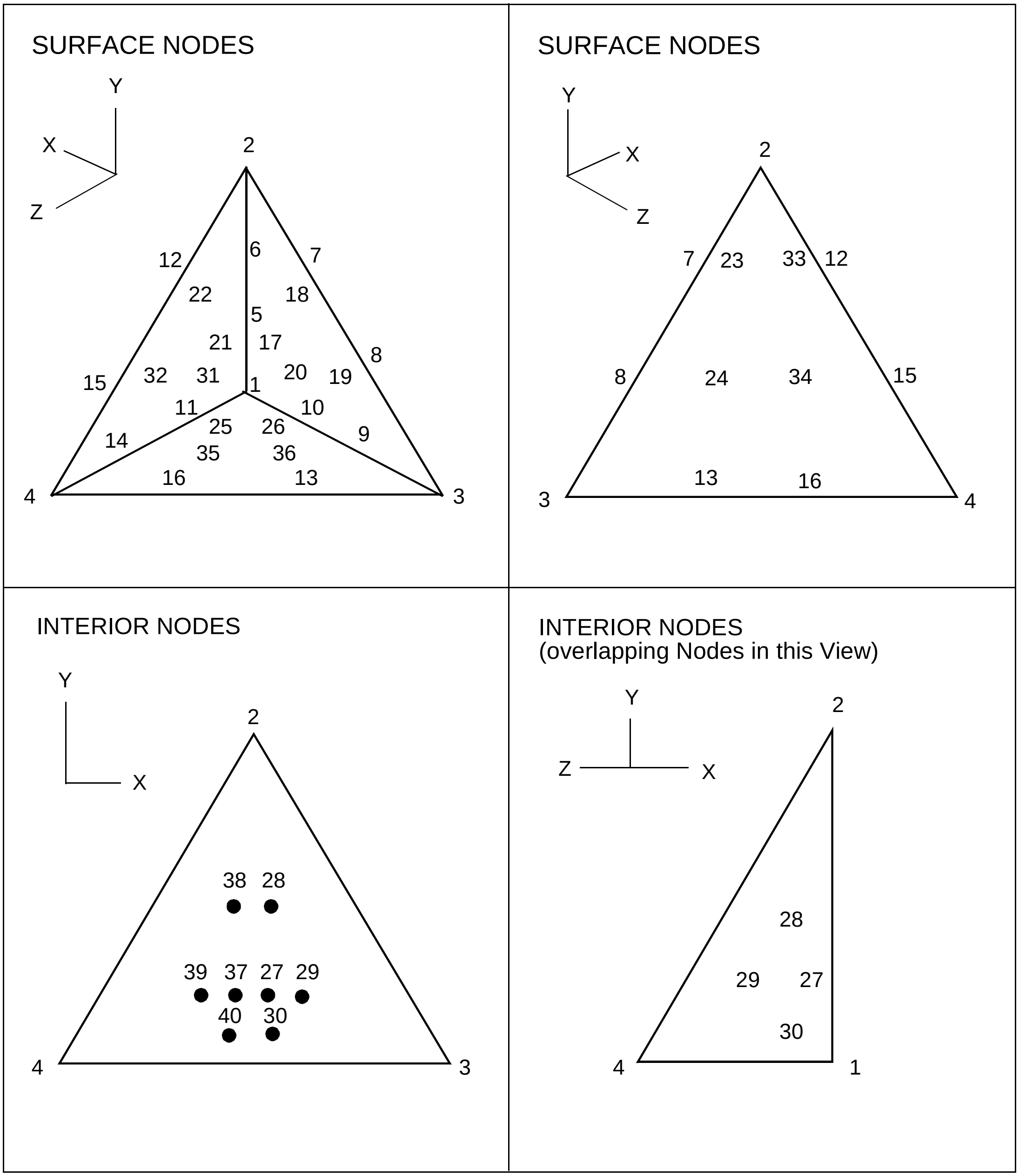


Quad16

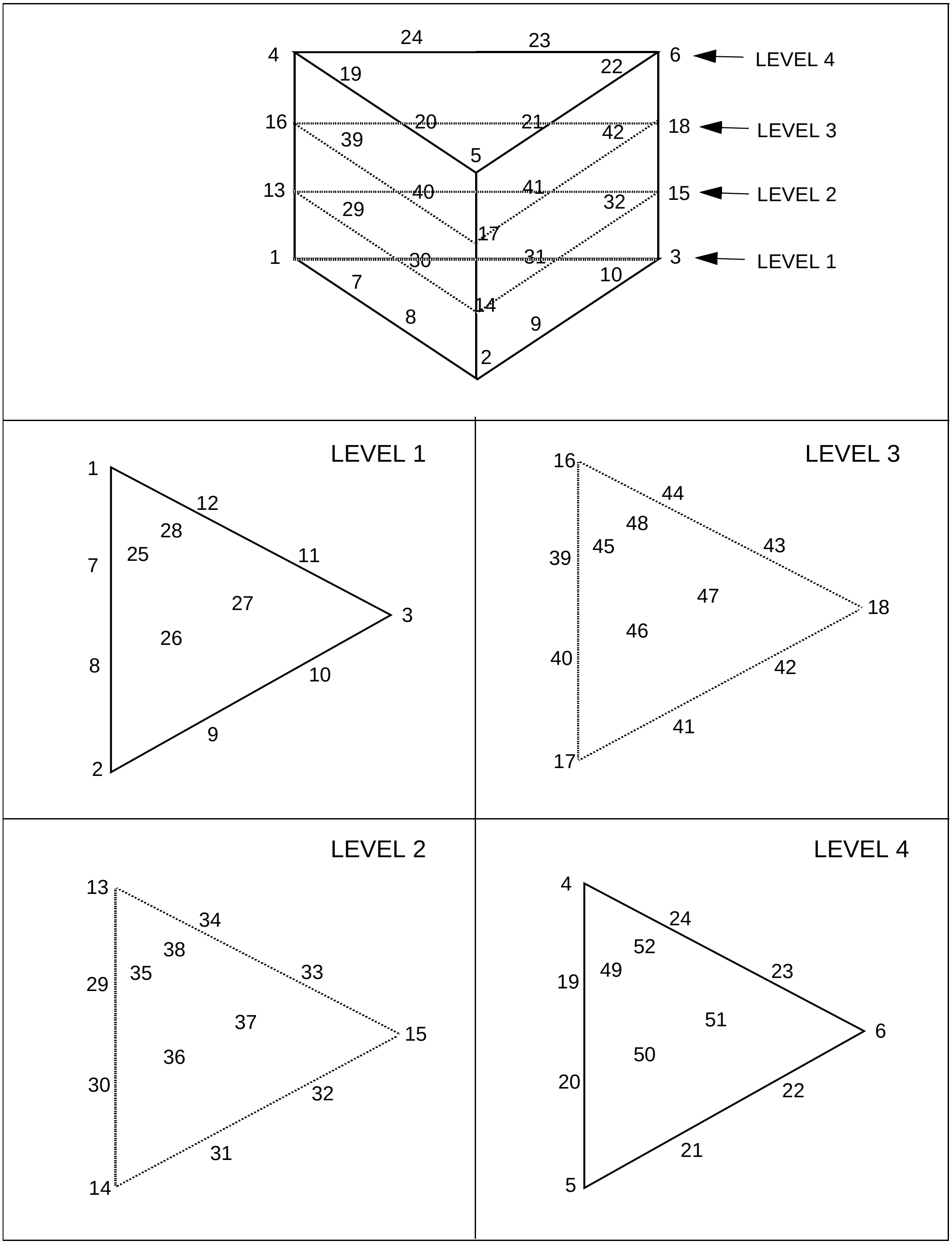


Tri13

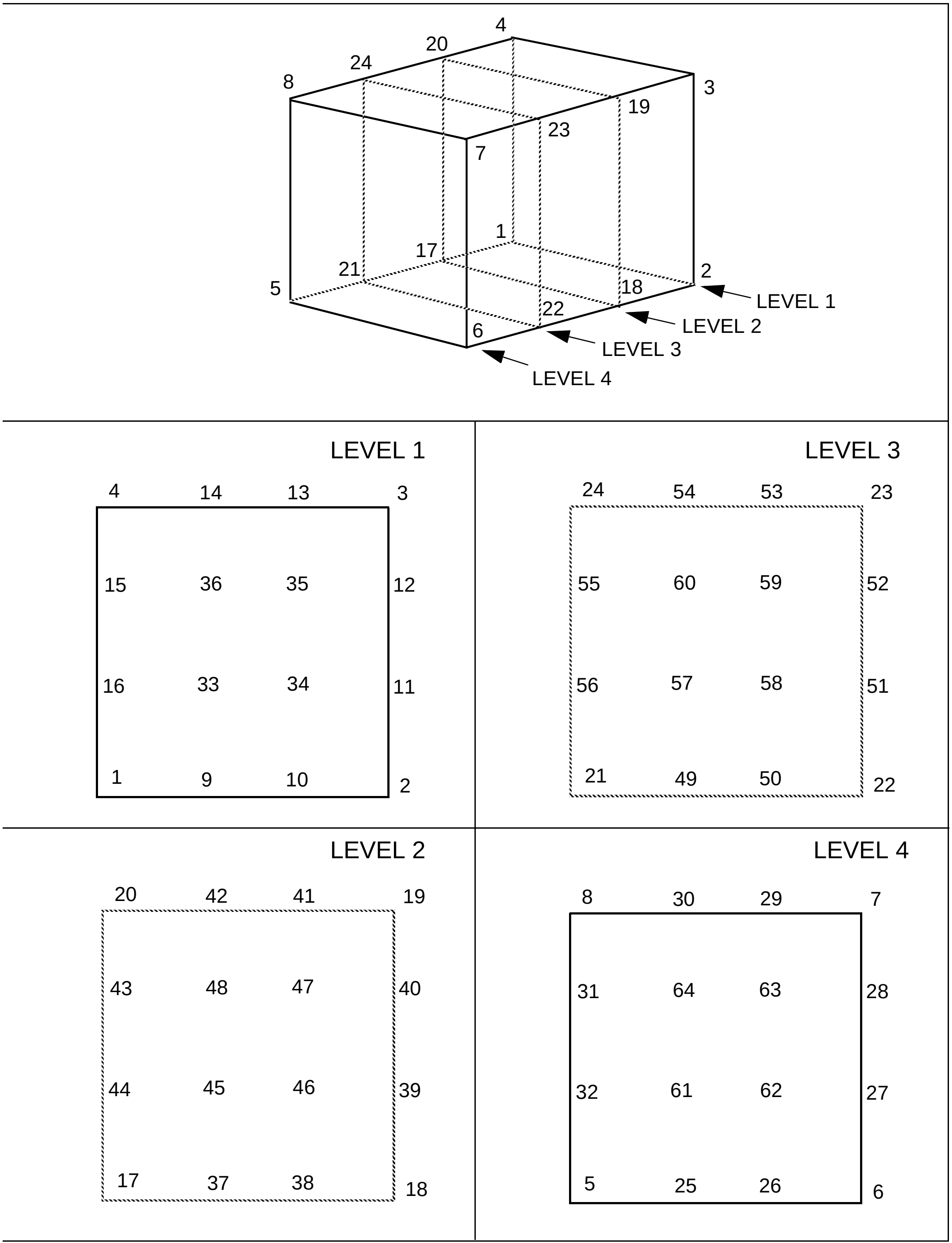
Tet40


SURFACE NODES
Y
X
Z
2
12
6
7
22
18
5
21
17
8
15
32
31
1
20
19
11
10
14
25
26
9
35
36
16
13
4
3
SURFACE NODES
Y
X
Z
2
7
23
33
12
8
24
34
15
13
16
3
4
INTERIOR NODES
Y
X
2
38
28
39
37
27
29
40
30
4
3
INTERIOR NODES
(overlapping Nodes in this View)
Y
Z
X
2
28
29
27
30
4
1

Wedge 52



Hex64


4
20
24
8
3
19
23
7
1
17
21
5
2
18
22
6
LEVEL 1
LEVEL 2
LEVEL 3
LEVEL 4
LEVEL 1
4 14 13 3
15 36 35 12
16 33 34 11
1 9 10 2
LEVEL 3
24 54 53 23
55 60 59 52
56 57 58 51
21 49 50 22
LEVEL 2
20 42 41 19
43 48 47 40
44 45 46 39
17 37 38 18
LEVEL 4
8 30 29 7
31 64 63 28
32 61 62 27
5 25 26 6

INDEX

MSC.Patran Reference Manual
Part 3: Finite Element Modeling

A

access finite element modeling, 5
analysis coordinate frame, 2
any, 306
arc center
    node, 82
arc method, 136
aspect ratio, 241
associate, 9
    curve, 165
    node, 168
    point, 164
    solid, 167
    surface, 166
associate action, 163
attributes, 2
Auto TetMesh, 8

B

bars, 287
building finite element model, 6

C

collapse, 249
connectivity, 2
constraint, 2
create
    mesh seeding, 19
    meshing curves, 14
    meshing solids, 17
    meshing surfaces, 15
    remeshing / reseeding, 21
create action, 12
Create Node
    edit, 80
creating finite element model, 8
curve method, 165
cyclic symmetry, 2, 115

D

degrees-of-freedom, 2, 110
delete, 9
    any, 306
    DOF List, 316
    element, 313
    mesh control, 311
    mesh curve, 309
    mesh seed, 307
    mesh solid, 310
    mesh surface, 308
    MPCs, 314
    node, 312
    superelement, 315
delete action, 305
dependent DOF, 2
disassociate, 171
    elements, 172
    node, 173
disassociate action, 171

E

edge angle, 247
edit, 279
    Create Node, 80
editing, 8
element, 313
    boundaries, 194
    connectivity, 197
    duplicates, 195
    IDs, 201
    Jacobian Ratio, 199
    Jacobian Zero, 200
    normals, 196
element attributes, 257
element coordinate system, 259
element topology, 13
element-element
    geometry fit, 198

elements, 130, 131, 132, 159, 186
  mirror, 132
  renumber, 159
  rotate, 131
  translate, 130
equivalence
  all, 179
  group, 180
  list, 181
equivalence action, 176
equivalencing, 2, 9
examples, 318
explicit, 2
extract method
  multiple nodes, 89
  node, 84
  single node, 87
extrude method, 137

F

Feature Select, 51, 55
FEM data, 151
finite element, 2
finite element model, 2
free edges, 2
free faces, 2

G

glide control, 139
glide method, 138
glide-guide control, 142
glide-guide method, 140
graphics, 111
group, 180

H

hex
  all, 226
  aspect, 228
  edge angle, 229
  face skew, 230
  face taper, 233
  face warp, 231
  twist, 232

I

implicit, 2
independent DOF, 2
interpolate method
  node, 91, 94
intersect method
  node, 96
IsoMesh, 2, 8, 15
  2 curve, 39
  curve, 38
  surface, 40

J

Jacobian Ratio, 3
Jacobian Zero, 3

L

library, 3
list, 181
loft method, 150

M

mesh, 268
  On Mesh, 49
mesh control, 75, 311
mesh control data, 152
mesh curve, 309
mesh paths, 15, 17
mesh seed, 30, 276, 307
  curvature based, 33
  one way bias, 31
  tabular, 34
  two way bias, 32
  uniform, 30
mesh seed attributes, 260
mesh seeding, 19
mesh solid, 310
mesh surface, 308
mesh transitions, 19
meshing curves, 14
meshing solids, 17
meshing surfaces, 15
midnode
  normal offset, 235
  tangent offset, 236

modify, 9
- bars, 287
- edit, 279
- mesh, 268
- mesh seed, 276
- MPCs, 301
- nodes, 297
- quads, 292
- trais, 288

modify action, 267
MPC, 3, 262
- cyclic symmetry, 115
- degrees-of-freedom, 110
- graphics, 111
- sliding surface, 116

MPC create, 9
MPC terms, 110
MPC types, 109, 113
MPCs, 109, 301, 314
multiple MPCs, 111
multiple nodes
- extract method, 89

N

node, 312
- extract method, 84
- IDs, 234
- interpolate method, 91, 94
- intersect method, 96
- offset method, 100
- pierce method, 103
- project method, 105

node distance, 256
node location, 255
nodes, 80, 125, 126, 128, 158, 186, 297
- mirror, 128
- renumber, 158
- rotate, 126
- translate, 125

non-uniform seed, 3
normal method, 143
normals, 3

O

offset method
- node, 100

optimization, 3
optimization method, 187
optimize, 9
- nodes / elements, 186

optimize action, 184

P

parameters, 3
paths, 3
paver, 3, 8, 15
pierce method
- node, 103

point method, 164
project method
- node, 105

Q

quad, 292
- all, 206
- aspect, 208
- skew, 210
- taper, 211
- warp, 209

R

radial cylindrical method, 144
radial spherical method, 145
reference coordinate frame, 3
remeshing / reseeding, 21
renumber, 3, 9
- action, 157

rezoning, 151

S

seeding, 3
seeding solid, 19
seeding surface, 19
shape, 3
Sheet Body, 53
show, 9
- element attributes, 257
- element coordinate system, 259
- mesh control attributes, 261
- mesh seed attributes, 260
- MPC, 262
- node distance, 256
- node location, 255

show action, 254

single node
    extract method, 87
skew, 238
sliding surface, 3, 116
solid
    IsoMesh, 42
    TetMesh, 45
solid method, 167
spherical theta method, 146
sub MPC, 3
surface method, 166
sweep, 9
    arc, 136
    extrude, 137
    glide, 138
    glide-guide, 140
    loft, 150
    normal, 143
    radial cylindrical, 144
    radial spherical, 145
    spherical theta, 146
    vector field, 148
sweep action, 135

T

taper, 246
term, 3
Tet
    all, 212
    aspect, 214
    collapse, 217
    edge angle, 215
    face skew, 216
TetMesh, 4, 8
    parameters, 47
theory, 238
    aspect ratio, 241
    collapse, 249
    edge angle, 247
    skew, 238
    taper, 246
    twist, 250
    warp, 245
topology, 4
transform, 9
transform action, 124
transitions, 4
tria
    all, 202
    aspect, 204
    skew, 205
triangular elements, 19
trias, 288
twist, 250
types, 4

U

uniform seed, 4

V

vector field method, 148
verification, 4, 9

verify
- element
  - boundaries, 194
  - connectivity, 197
  - duplicates, 195
  - IDs, 201
  - Jacobian Ratio, 199
  - Jacobian Zero, 200
  - normals, 196
- element-element
  - geometry fit, 198
- hex
  - all, 226
  - aspect, 228
  - edge angle, 229
  - face skew, 230
  - face taper, 233
  - face warp, 231
  - twist, 232
- midnode
  - normal offset, 235
  - tangent offset, 236
- node
  - IDs, 234
- quad
  - all, 206
  - aspect, 208
  - skew, 210
  - taper, 211
  - warp, 209
- Tet
  - all, 212
  - aspect, 214
  - collapse, 217
  - edge angle, 215
  - face skew, 216
- tria
  - all, 202
  - aspect, 204
  - skew, 205
- wedge
  - all, 218
  - aspect, 220
  - edge angle, 221
  - face skew, 222
  - face taper, 225
  - face warp, 223
  - twist, 224

verify action, 190

W

warp, 245

wedge
- all, 218
- aspect, 220
- edge angle, 221
- face skew, 222
- face taper, 225
- face warp, 223
- twist, 224