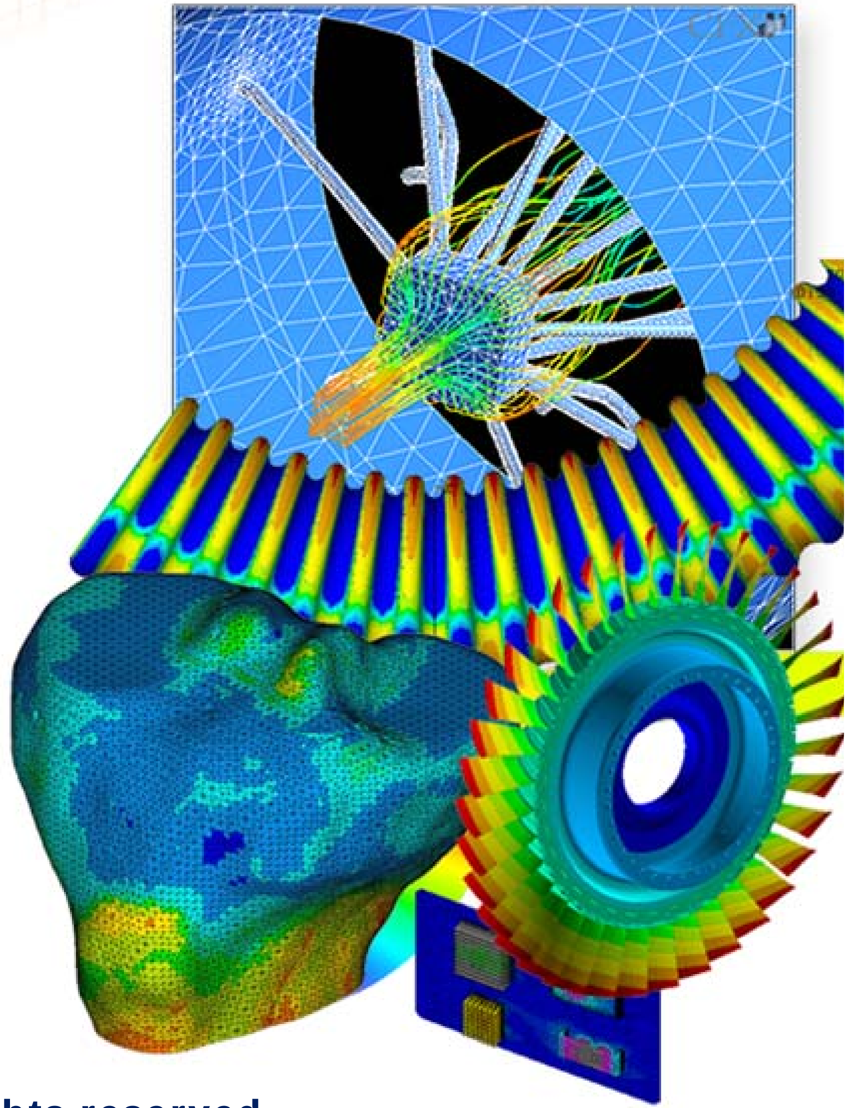
ENGINEERING ADVANTAGE

ANSYS Element Selection eLearning



Peter Barrett

October 2012

ANSYS Element Selection

- What is the best element type(s) for my analysis?
  - **Best Answer - It depends!**

- **What is the goal of the analysis?**
  - Displacements, temperatures, mode shapes, nominal strength, fatigue life, stresses from fluid structure interaction, etc.

- **What are the shapes of my parts/assembly?**
  - What types of symmetry can I take advantage of?
  - Is it long and slender where beams might be useful?
  - Thin walled where Shell or Solid-Shell elements can be used?
  - Should I model the full assembly in one shot or use submodeling and/or substructuring?

- **What type of computational requirements are needed?**
  - Linear vs. Nonlinear, Static vs. Dynamic, Time Domain vs. Freq. Domain?
    - The more complex the analysis, the fewer elements can be effectively solved

ANSYS V11 – 186 Elements Available!

Element Library
LINK1
BEAM3
BEAM4
SOLID5
COMBIN7
LINK8
INFIN9
LINK10
LINK11
CONTAC12
PLANE13
COMBIN14
PIPE16
PIPE17
PIPE18
PIPE20
MASS21
BEAM23
BEAM24
PLANE25
MATRIX27
SHELL28
FLUID29
FLUID30
LINK31
LINK32
LINK33
LINK34
PLANE35
SOURC36
COMBIN37
FLUID38
COMBIN39
COMBIN40

SHELL41
PLANE42
SHELL43
BEAM44
SOLID45
SOLID46
INFIN47
MATRIX50
CONTAC52
PLANE53
BEAM54
PLANE55
SHELL57
PIPE59
PIPE60
SHELL61
SOLID62
SHELL63
SOLID65
PLANE67
LINK68
SOLID69
SOLID70
MASS71
PLANE75
PLANE77
PLANE78
FLUID79
FLUID80
FLUID81
PLANE82
PLANE83
SOLID87
VISCO88
VISCO89
SOLID90

SHELL91
SOLID92
SHELL93
CIRCU94
SOLID95
SOLID96
SOLID97
SOLID98
SHELL99
VISCO106
VISCO107
VISCO108
TRANS109
INFIN110
INFIN111
INTER115
FLUID116
SOLID117
HF118
HF119
HF120
PLANE121
SOLID122
SOLID123
CIRCU124
CIRCU125
TRANS126
SOLID127
SOLID128
FLUID129
FLUID130
SHELL131
SHELL132
FLUID136
FLUID138
FLUID139

FLUID141
FLUID142
ROM144
PLANE145
PLANE146
SOLID147
SOLID148
SHELL150
SURF151
SURF152
SURF153
SURF154
SURF156
SHELL157
LINK160
BEAM161
PLANE162
SHELL163
SOLID164
COMBI165
MASS166
LINK167
SOLID168
TARGE169
TARGE170
CONTA171
CONTA172
CONTA173
CONTA174
CONTA175
CONTA176
CONTA177
CONTA178
PRETS179
LINK180
SHELL181

PLANE182
PLANE183
MPC184
MPC184-Link/Beam
MPC184-Slider
MPC184-Revolute
MPC184-Universal
MPC184-Slot
MPC184-Point
MPC184-Translation
MPC184-Cylindrical
MPC184-Planar
MPC184-Weld
MPC184-Orient
MPC184-Spherical
MPC184-General
SOLID185
SOLID186
SOLID187
BEAM188
BEAM189
SOLSH190
SOLID191
INTER192
INTER193
INTER194
INTER195
MESH200
FOLLW201
INTER202
INTER203
INTER204
INTER205
SHELL208
SHELL209
COMBI214

COMBI214
PLANE223
SOLID226
SOLID227
PLANE230
SOLID231
SOLID232
SURF251
SURF252
REINF265
SHELL281

Note:
Some Beam, Pipe, Spar, Shell, Solid elements were removed or changed to legacy after this release.

Although undocumented these elements remain in the code but require users to have access to V11 help or older documentation to use.

ANSYS V14 – 166 Elements Available!

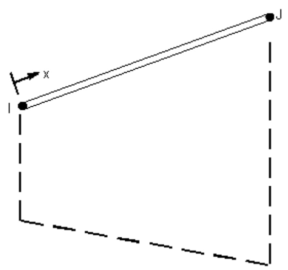
CAE ASSOCIATES

- SOLID5
- INFIN9
- LINK11
- PLANE13
- COMBIN14
- MASS21
- PLANE25
- MATRIX27
- SHELL28
- FLUID29
- FLUID30
- LINK31
- LINK33
- LINK34
- PLANE35
- SOURC36
- COMBIN37
- FLUID38
- COMBIN39
- COMBIN40
- SHELL41
- INFIN47
- MATRIX50
- PLANE53
- PLANE55
- SHELL61
- SOLID62
- SOLID65
- LINK68
- SOLID70
- MASS71
- PLANE75
- PLANE77
- PLANE78
- FLUID79
- FLUID80
- FLUID81
- PLANE83
- SOLID87
- SOLID90
- CIRCU94
- SOLID96
- SOLID97
- SOLID98
- INFIN110
- INFIN111
- INTER115
- FLUID116
- HF118
- HF119
- HF120
- PLANE121
- SOLID122
- SOLID123
- CIRCU124
- CIRCU125
- TRANS126
- FLUID129
- FLUID130
- SHELL131
- SHELL132
- FLUID136
- FLUID138
- FLUID139
- FLUID141
- FLUID142
- ROM144
- SURF151
- SURF152
- SURF153
- SURF154
- SURF156
- SHELL157
- SURF159
- LINK160
- BEAM161
- PLANE162
- SHELL163
- SOLID164
- COMBI165
- MASS166
- LINK167
- SOLID168
- TARGE169
- TARGE170
- CONTA171
- CONTA172
- CONTA173
- CONTA174
- CONTA175
- CONTA176
- CONTA177
- CONTA178
- PRETS179
- LINK180
- SHELL181
- PLANE182
- PLANE183
- MPC184
- MPC184-Link/Beam
- MPC184-Slider
- MPC184-Revolute
- MPC184-Universal
- MPC184-Slot
- MPC184-Point
- MPC184-Translatior
- MPC184-Cylindrical
- MPC184-Planar
- MPC184-Weld
- MPC184-Orient
- MPC184-Spherical
- MPC184-General
- MPC184-Screw
- SOLID185
- SOLID186
- SOLID187
- BEAM188
- BEAM189
- SOLSH190
- INTER192
- INTER193
- INTER194
- INTER195
- MESH200
- FOLLW201
- INTER202
- INTER203
- INTER204
- INTER205
- SHELL208
- SHELL209
- CPT212
- CPT213
- COMBI214
- CPT215
- CPT216
- CPT217
- FLUID220
- FLUID221
- PLANE223
- SOLID226
- SOLID227
- PLANE230
- SOLID231
- SOLID232
- PLANE233
- SOLID236
- SOLID237
- HSFLD241
- HSFLD242
- SURF251
- SURF252
- REINF263
- REINF264
- REINF265
- SOLID272
- SOLID273
- SOLID278
- SOLID279
- SHELL281
- SOLID285
- PIPE288
- PIPE289
- ELBOW290
- USER300

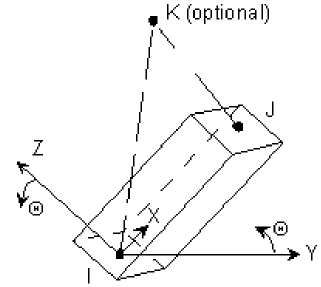
Note:
Many new elements have been added since V11 including new surface effect and advanced piping elements

Element Types define the active DOF

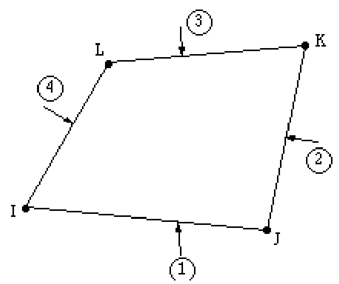
- The DOFs at a node are a function of the element type connected to the node.
  - *Be careful about the number of degrees of freedom generated in large beam/shell models since 6 degrees of freedom are created at each node*



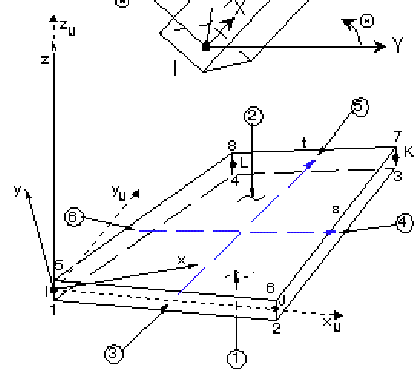
3-D Spar (Pin Joints)
UX, UY, UZ



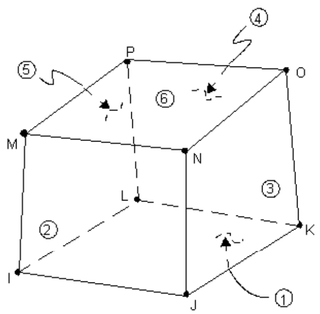
3-D Beam
UX, UY, UZ, ROTX, ROTY, ROTZ



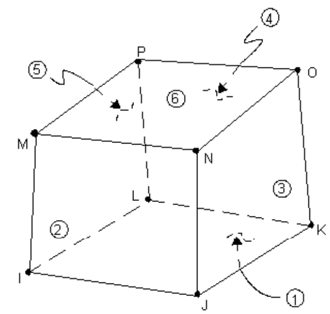
2-D or Axisymmetric Solid
UX, UY



3-D Quadrilateral Shell
UX, UY, UZ, ROTX, ROTY, ROTZ



3-D Structural Solid
UX, UY, UZ



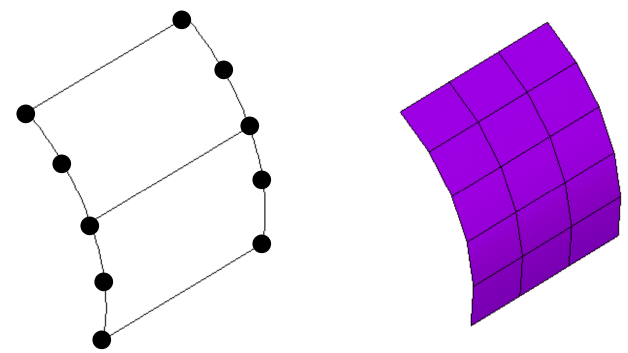
3-D Thermal Solid
TEMP

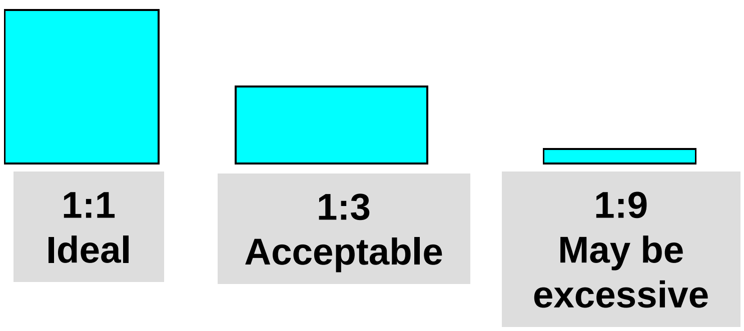
Element Shape Function

- FEA solves for DOF values **only** at nodes.
- An element **shape function** is the “shape” of the results within the element.
- Most elements fall within two categories:
  - Linear or lower-order elements:
    - Corner or end nodes only.
    - Assume a linear variation of DOF values within element.
      - (Using enhanced strain for quad/brick shapes will adds extra shapes for bending)
    - Only allows straight sides. Curves are faceted.
  - Quadratic or higher-order:
    - Corner or end nodes and mid-side nodes.
    - Assume a quadratic variation of DOFs within element.
    - Allows for modeling of curved boundaries

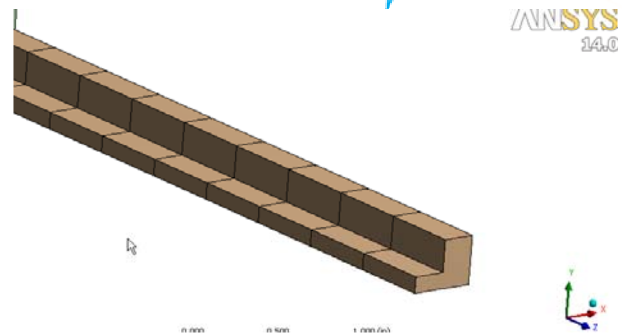
Guidelines for Element Selection in Meshing

- Element Usage
  - For curved surfaces use higher order curved elements, especially for contact problems
  - Quadrilateral elements are preferred over triangular elements, particularly for well-shaped geometry.
  - For very large strain problems using triangles can prevent element distortion
  - Avoid high aspect ratio of the element sides.
  - Need to also consider the gradient across the element to determine the accuracy of the response



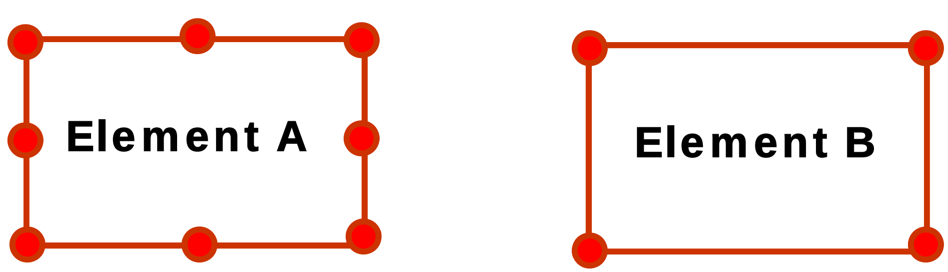


- Beam Elements:
  - Meshing of Line Bodies or Lines or through direct node and element generation
  - Using a third node or orientation key point to orient
  - View "/ESHAPE" to check the X-sect orientation
- Shell Elements:
  - Meshing of surface bodies / Areas
  - Element types controlled by assignment or advanced mesh settings. Use consistent normals
- Solid Element Midside Nodes:
  - Program controlled (default), dropped, or kept.



Details of "Mesh"

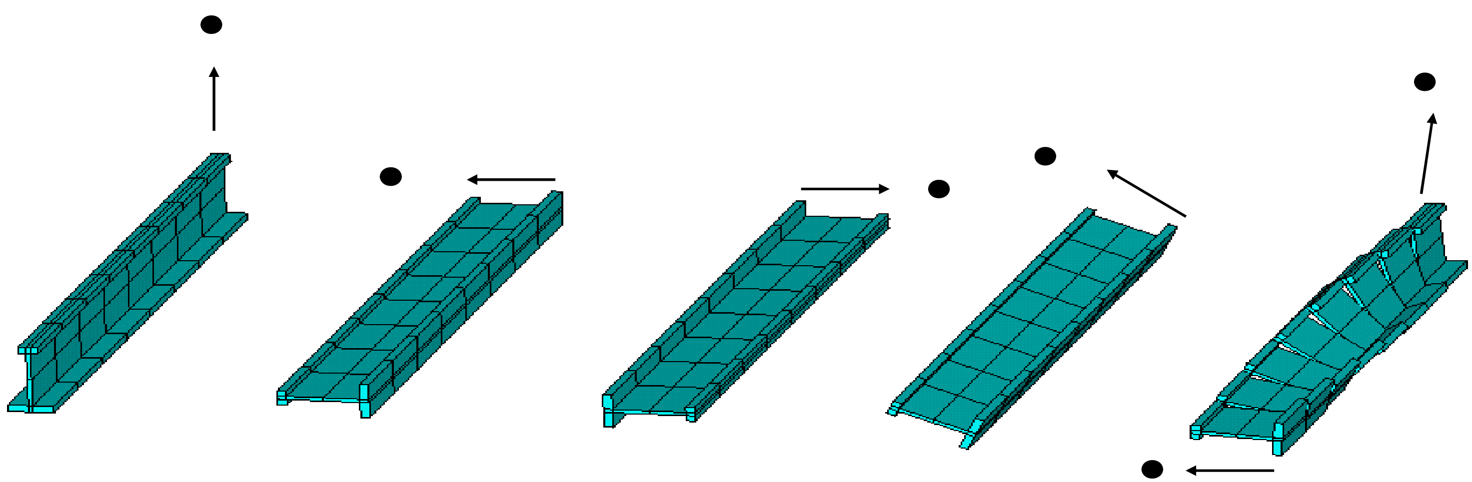
Defaults	
Physics Preference	Mechanical
Relevance	100
Advanced	
Relevance Center	Coarse
Element Size	Default
Shape Checking	Standard Mechanical
Solid Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Initial Size Seed	Active Assembly
Smoothing	Low
Transition	Fast
Statistics	
Nodes	9042
Elements	4962



Kept Dropped

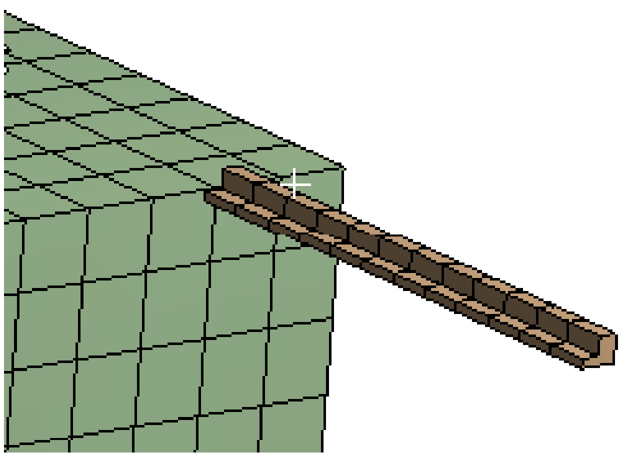
Why Use Beam Elements?

- Advantages:
  - Fast to solve, Follow Beam Theory, Less results data, Forces and moments and linearized stresses directly available
- Disadvantages:
  - Cannot account for local stress concentrations, More difficult to define, Limited to constant or linear tapered cross-sections
- Common Errors
  - Cross-Section not oriented correctly
  - Insufficient constraints when used in conjunction with shell and solid elements



Why Use Beam Elements?

- Common Modeling Issues
  - BEAM 188: KEYOPT(3) Shape functions along the length:
    - 0 -- Linear (default) *Recommend either 2 -- Quadratic or 3 -- Cubic (beam theory)*
    - Mechanical defaults to Keyopt (3) = 2
  - BEAM 188/189 Section Controls
    - The # of integration points in each cross section can greatly influence solution time and results file size **(Main Menu>Preprocessor>Sections>Beam>Common Sections > secdata)**
    - For linear materials use a coarse cross-section mesh; to capture plasticity a refined cross section mesh might be needed.
  - Model connections correctly (Fully fixed vs. pinned vs. partial moment release)



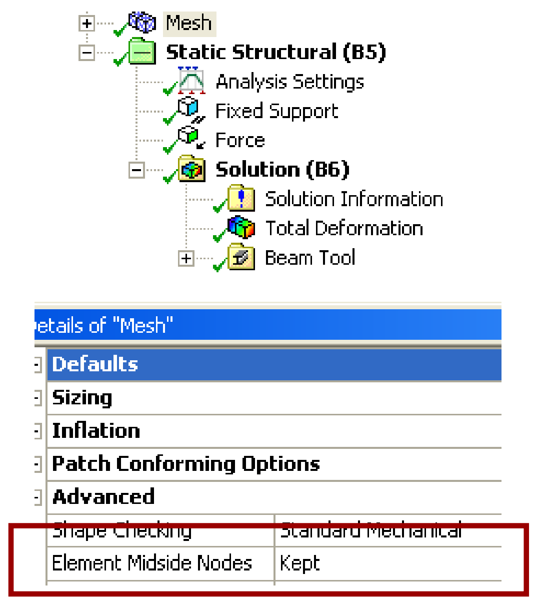
```
*** ERROR ***                           CP =       4.984   TIME= 09:36:36
A small negative equation solver pivot term has been encountered at the
ROTZ degree of freedom of node 2870.  Check for an insufficiently
constrained model.


NUMBER OF WARNING MESSAGES ENCOUNTERED=          2
NUMBER OF ERROR   MESSAGES ENCOUNTERED=          1
```

ANSYS Beam Elements Tips

- Lower Order Beam Elements (188)
  - Mechanical Default
  - Recommended for straight beams and for combined beam/shell models when connected to lower order shells and solids (Shell 181, Solid185 etc)

- Higher Order Beams (189) are recommended for curved geometry and for combined meshes with higher order shells/solids
  - Use Mesh Details > Advanced Settings to Force Higher Order Beams (189) in Mechanical

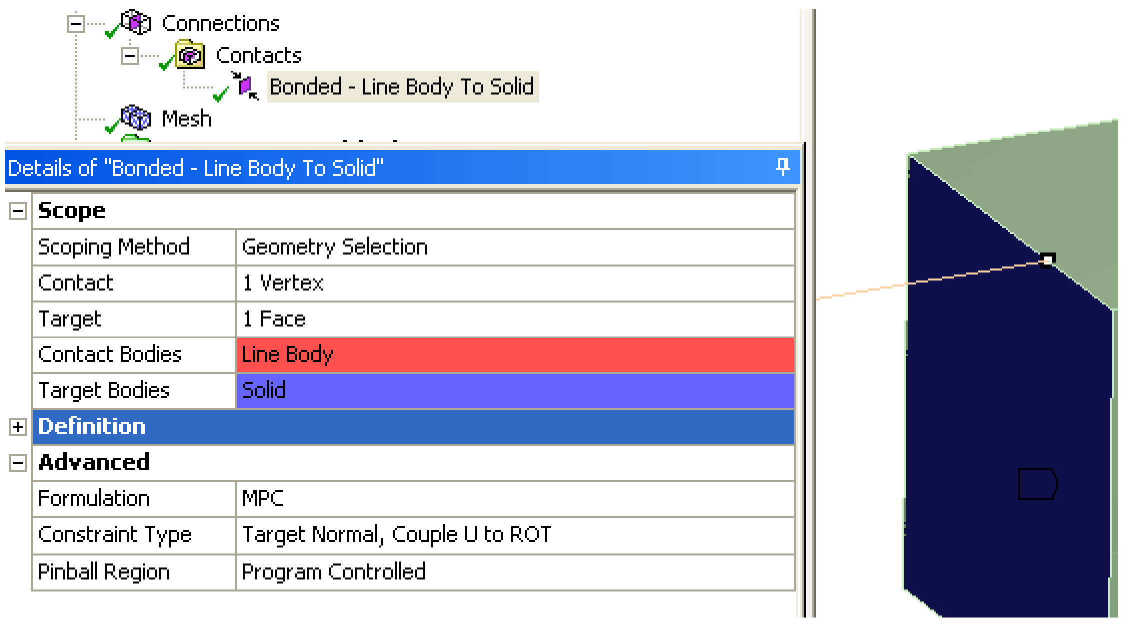
```
finish
/prep7
keyopt,b_188,3,3
finish
/solu
```



ANSYS Beam Elements Tips

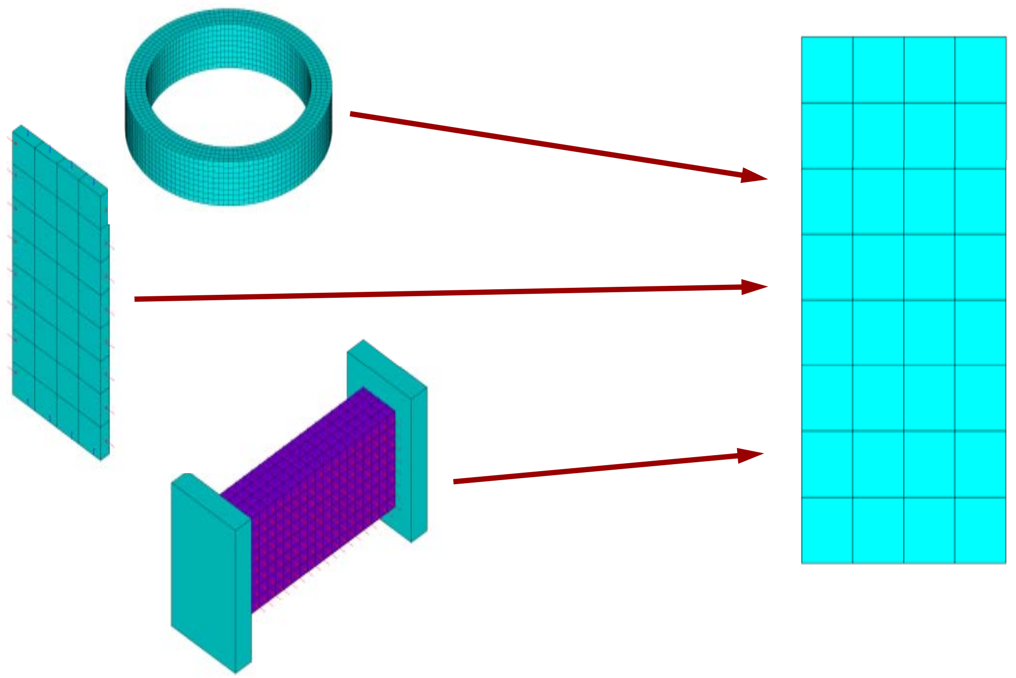
- Mechanical Spot Weld feature will automatically create beams and “spiders” to spread the loading between bodies

- Avoid connecting single beams directly to solids
  - Use MPC contact, Constraint Equations or spokes to distribute the load
  - Spoke elements with lengths equal to half the member section is an preferred modeling approach.

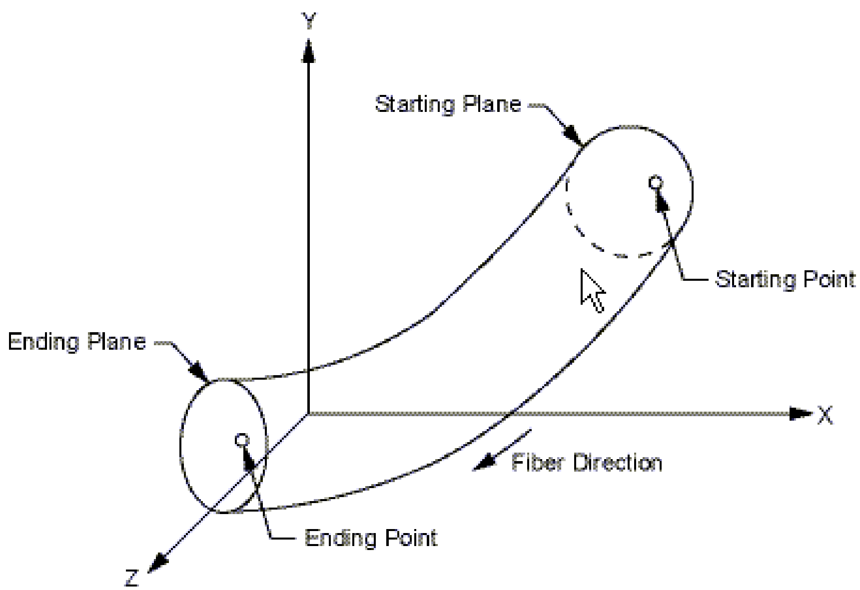


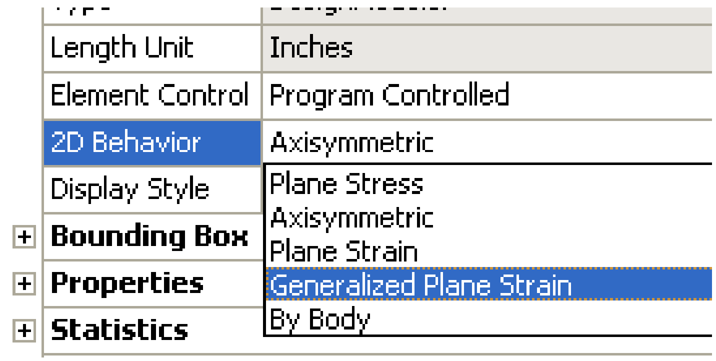
Why 2-d or Axisymmetric Solid Elements?

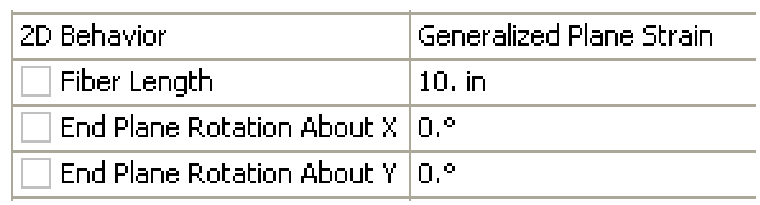
- Advantages:
  - Faster to Solve, Less data created, Easy to check / Evaluate stresses
- Disadvantages:
  - Creating the 2-d surface and correct orientation
  - Modeling approximations of 3-d features
- Common Errors
  - Not modeling @ z=0 with positive X for Radius
  - Selecting incorrect formulation (Plane Stress vs. Pl. Strain vs. Axisymmetric)



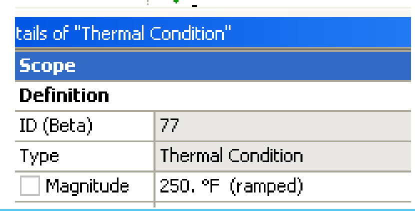
- Generalized Plane Strain
  - A finite deformation domain length in the z direction (Z direction strain is not required to be zero)
  - Simulate 3-D deformations using 2-D options. Example – Section of a cylinder or long body under uniform Temp. Added Commands – (GSBDATA, GSGDATA, GSSOL,GSLIST)





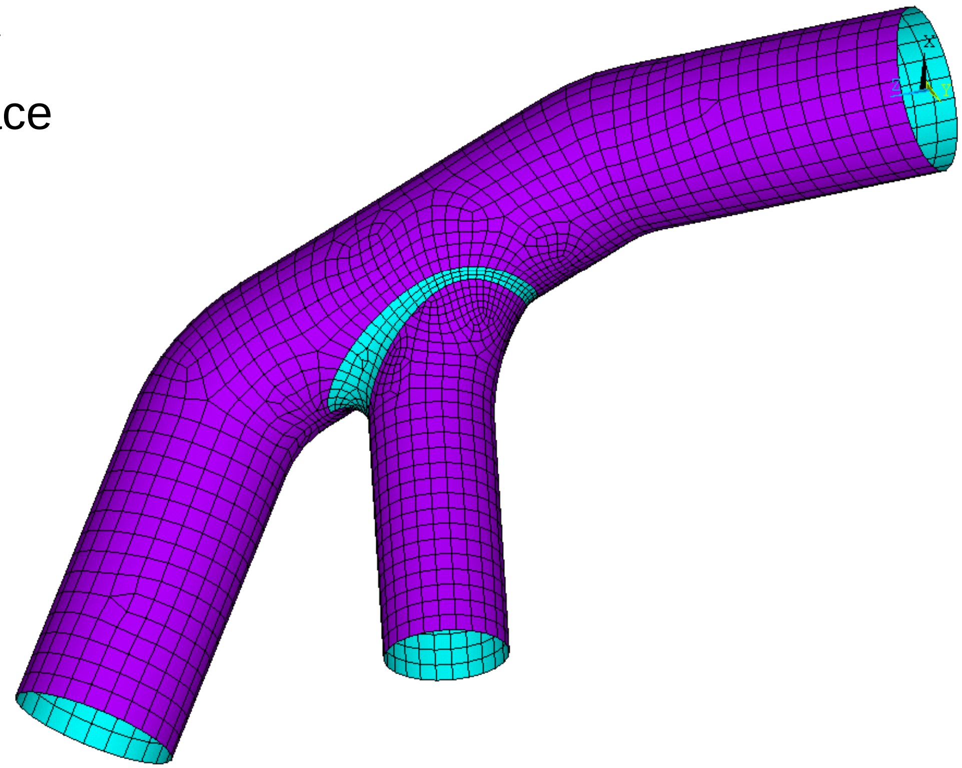


- “Free” Thermal Expansion under Thermal Loading



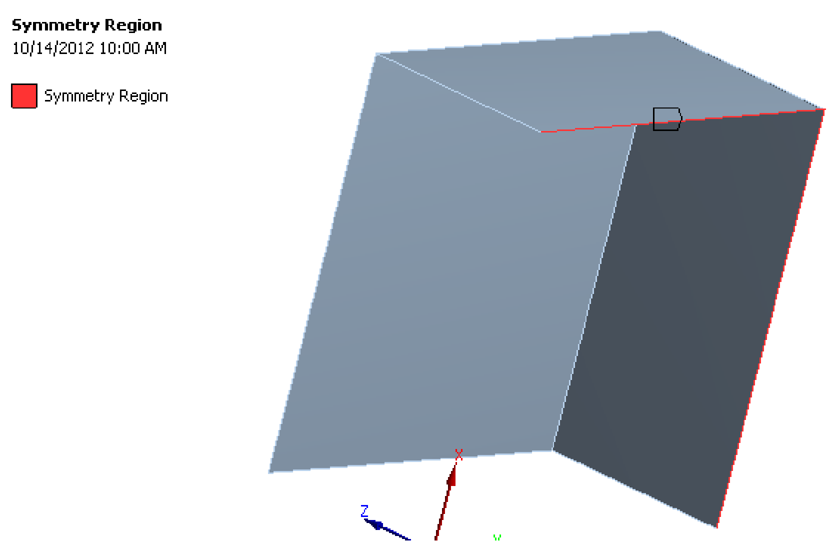
Why Use Shell Elements?

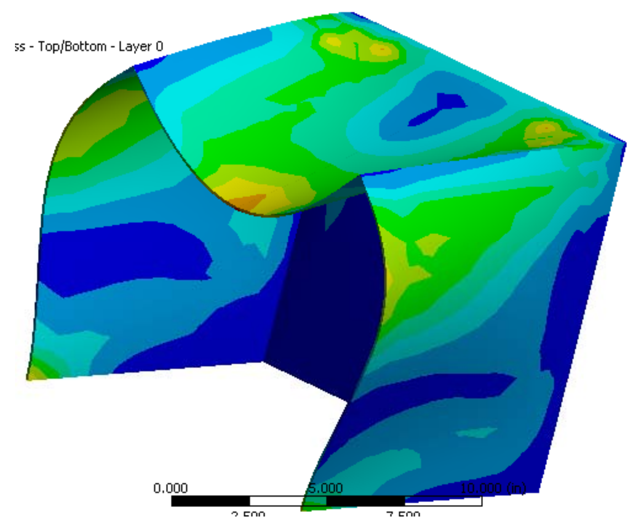
- Advantages:
  - Faster to Solve, No length to thickness requirements, Less data created, Forces and Moments easily obtained
- Disadvantages:
  - Difficult to define contact, normal's and connections
- Common Errors
  - Defining normal's inconsistently
  - Postprocessing the wrong surface



Why Use Shell Elements?

- Common Modeling Issues
  - SHELL188: KEYOPT(3) Integration option:
    - 0 -- Reduced integration with hourglass control (default)
    - 2 -- Recommend Full integration with incompatible modes
    - Mechanical defaults to Shell 181 - Keyopt (3) = 2
  - Symmetry Boundary Conditions
    - Make sure to also constrain the in-plane rotations for symmetric surfaces
  - Turn Large Deflection On
    - For Pressure loads large deflection is often required since membrane stiffness is not accounted for with small deflections

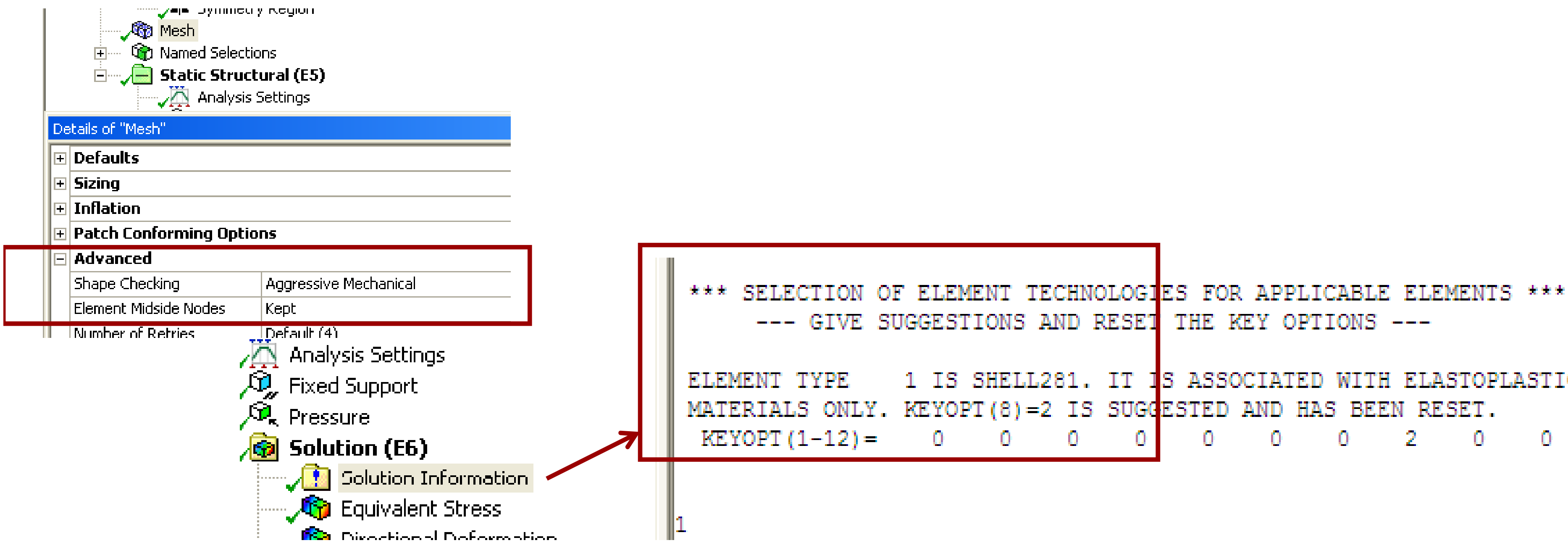




Shell Elements Tips

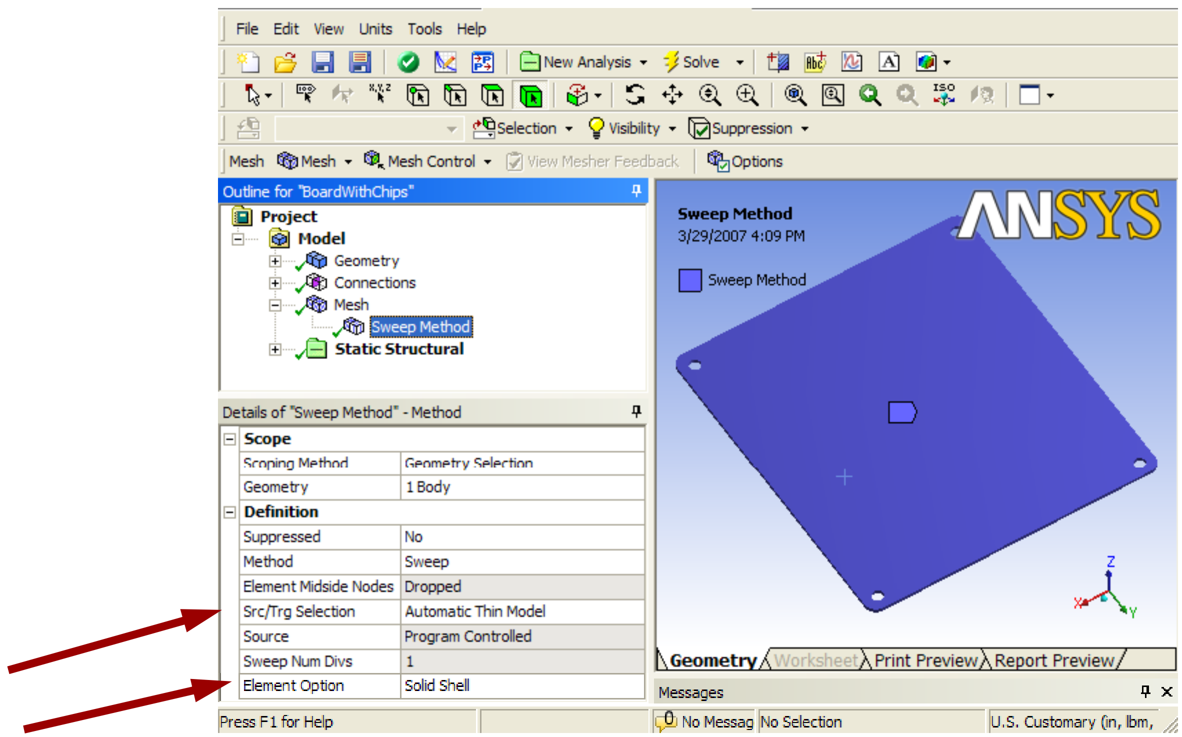
- Lower Order Shell Elements (181) Mechanical Default
  - Recommended for flat bodies and for combined beam/shell models when connected to lower order beams and solids (Beam 188, Solid185 etc)

- Higher Order Shells (281) are recommended for curved geometry and for combined meshes with higher order beams/solids
  - Use Mesh Details > Advanced Settings to Force Higher Order Shells (281) in Mechanical



Solid Shell Elements

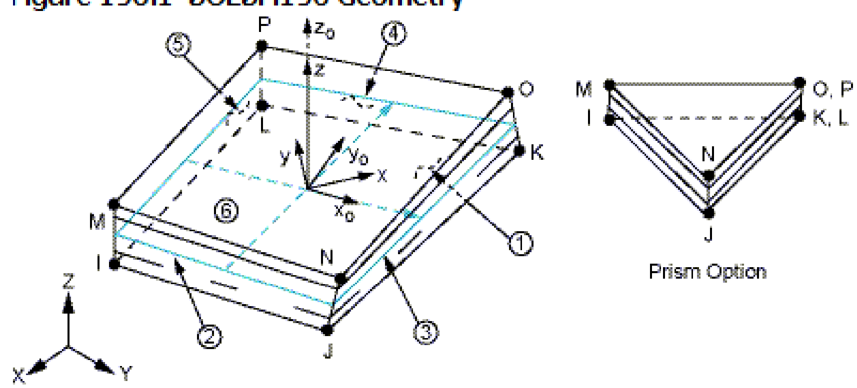
- Solid Shell (SOLSH190) elements can be used for thin solids in Mechanical.
  - Set the mesh method to Sweep.
  - Choose Src/Tg Selection: “Automatic Thin Model”.
  - Element Option: “Solid Shell”.



Why Use Solid Shell Elements?

- Advantages:
  - Faster to Solve, No length to thickness requirements, Thickness defined by geometry, Auto Contact generation available, Easy transition to standard 3-d elements, Automatically creates tapered elements
- Disadvantages:
  - Sweepable Geometry and correct Element Orientation required
- Common Errors
  - Defining normal's inconsistently. “Thin Section” must be in element Z direction
- Common Modeling Issues
  - Lower Order Elements require refined mesh to modal small radii geometry
  - Meshing of complex geometry can be difficult

Figure 190.1 SOLSH190 Geometry

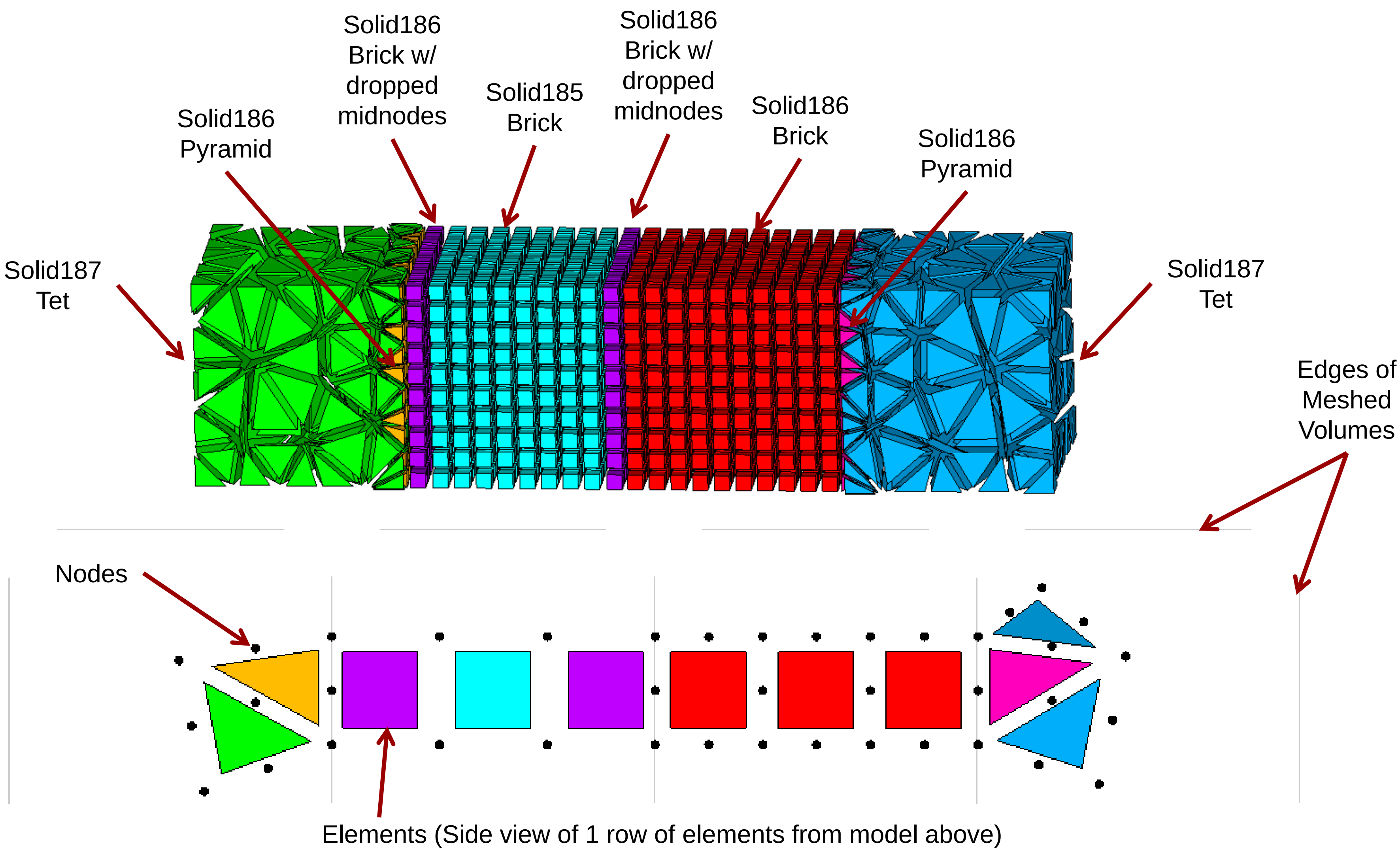


Why Use Solid Elements?

- Advantages:
  - Easy to Model, Provides 3-d geometry with results, Can include stress concentrations
- Disadvantages:
  - Computationally expensive, For thin and slender bodies may require excessive element count
- Common Errors
  - Using Singular results – stresses at sharp corners, fixed supports, point loads
- Common Modeling Issues
  - SOLID185 (Lower Order Brick): KEYOPT(3) Element technology:
    - 0 -- Full integration with B-BAR method (default)
    - 1 -- Uniform reduced integration with hourglass control
    - 2 -- Enhanced strain formulation – Recommended for Bending Problems
    - 3 -- Simplified enhanced strain formulation
    - Mechanical defaults to Keyopt (3) = 0 when midside nodes are dropped

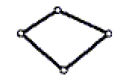
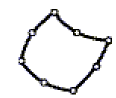
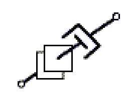
ANSYS Mechanical Mixed Order Meshing

- Mixed Order mesh transitions



Contact Elements

- Contact Element Types
  - 2D Surface-to-Surface
    - Plane Stress, Strain, Axisymmetric Elements
    - Rigid or Flex. Bodies
  - 3D Surface-to-Surface
    - Shell, Solid,-shell, Solid Elements
    - Rigid or Flex. Bodies
  - Point-to-Surface
    - Beams to shells or solids
  - Line-to-Line
    - Beam-to-Beam
    - Bending or solid-to-hollow pipe
  - Line-to-Surface
    - Beam or Shell edge to Solid
  - Node-to-Node
    - Specialty elements

CONTAC Elements	Graphic Pictorials
CONTA171 2-D 2-Node Surface-to-Surface Contact 2 nodes 2-D space DOF: UX, UY, TEMP, VOLT, AZ	
CONTA172 2-D 3-Node Surface-to-Surface Contact 3 nodes 2-D space DOF: UX, UY, TEMP, VOLT, AZ	
CONTA173 3-D 4-Node Surface-to-Surface Contact 4 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG	
CONTA174 3-D 8-Node Surface-to-Surface Contact 8 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG	
CONTA175 2-D/3-D Node-to-Surface Contact 1 node 2-D/3-D space DOF: UX, UY, UZ, TEMP, VOLT, AX, MAG	
CONTA176 3-D Line-to-Line Contact 3 nodes 3-D space DOF: UX, UY, UZ	
CONTA177 3-D Line-to-Surface Contact 3 nodes 3-D space DOF: UX, UY, UZ	
CONTA178 3-D Node-to-Node Contact 2 nodes 3-D space DOF: UX, UY, UZ	

Specialty Elements

- Lumped Mass - Mass21
  - Keyopt defines DOF (6 by default)
    - Set Keyopt 3=2 to reduce DOF if rotatry interia is not required
    - Make sure to specify Mass in all three directions
- Multi-Point Constraint - MPC184
  - Link, Slider, Revolute, Universal, Slot, Point-in-plane, Pin, Cylindrical, Weld, Spherical, Screw, etc.
    - Large deflection response
    - Automated Generation in WB
- Bolt Pretension – PRETS179
  - Small deflection Only
  - Use MPC184 or contact surface offset for large deflection
- Spring - (Dashpot) Elements
  - Combin14, 37, 39, 40, 214
    - Original Spring Elements
  - Combin214
    - Unsymmetrical Stiffness/Damping