

Nonlinear Contact Analysis Techniques using ANSYS

Mechanics Development Group



Outline of Presentation



- General considerations
- Contact applications
- Contact kinematics
- Node-to-node element **CONTA178**
- Node-to-surface element **CONTA175**
- Surface-to-surface elements **CONTA171-174**
- Multi-physics contact
- Bolt Pretension Element **PRETS179**
- Troubleshooting
- Conclusions
- Future developments

General Considerations in Contact Analysis



- The general goal for contact analysis is to determine
 - Contact stresses transmitted across contacting interface
 - Contacting area
- Contact problems present significant difficulties
 - Unknown contacting zone prior to the analysis
 - Contact constraint is either active or inactive
 - Friction introduces another kind of nonlinearities
 - A small amount of positive or negative relative sliding can change the sign of frictional forces/stresses completely.

General Considerations in Contact Analysis



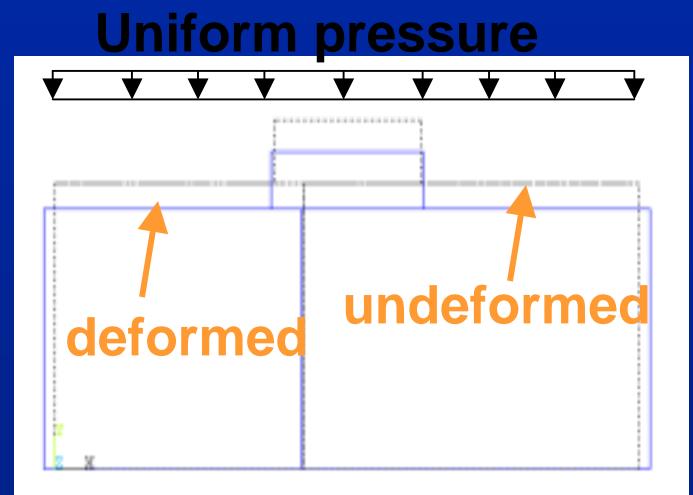
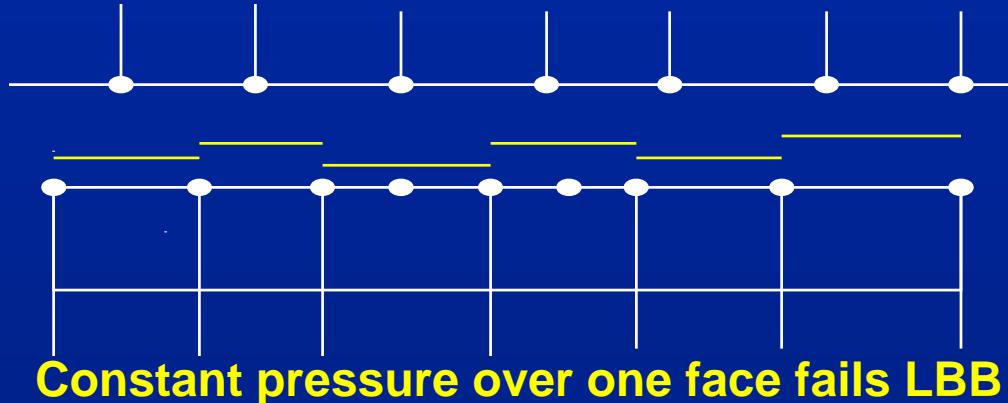
- **Contact finite element development covers**
 - Kinematics, discretization, Inequality
 - Accuracy, robustness and computational overhead
- **The numerical treatment of contact problems involves**
 - Formulation of geometry
 - Integration of interface laws
 - Variational formulation
 - Development of algorithms

- Contact problem involves a variety of geometric and kinematic situations
- Contact surface discretization
 - Contact surface must be discretized because the underlying bodies are discretized.
 - Node-node, node-surface, surface-surface
 - Smoothing provides a significant improvement in convergence behavior.
- Contact detection and searching
 - Global search, local search
- Penetration/gap calculation
 - Numerical iterations for higher order

General Considerations

ANSYS

- The resulting elements should be able to
 - Pass patch test
 - Mesh discretization effects
 - Satisfy Ladyshenskaja-Babuska-Brezzi (LBB) condition
 - Overconstraint criterion
 - Support contact with quadratic order element
 - Solve multi-field contact problems



Solving Larger Assembly Models



- In the early days of FE technology only the separate structural components of a structure were analyzed.
- It is now recognized that the interaction between structural parts can have a great influence on the results.
- Increased computer performance in combination with efficient solver technology and parallel computing techniques has resulted in FE models which may exceed 1,000,000 elements not only for linear but also for nonlinear analysis.

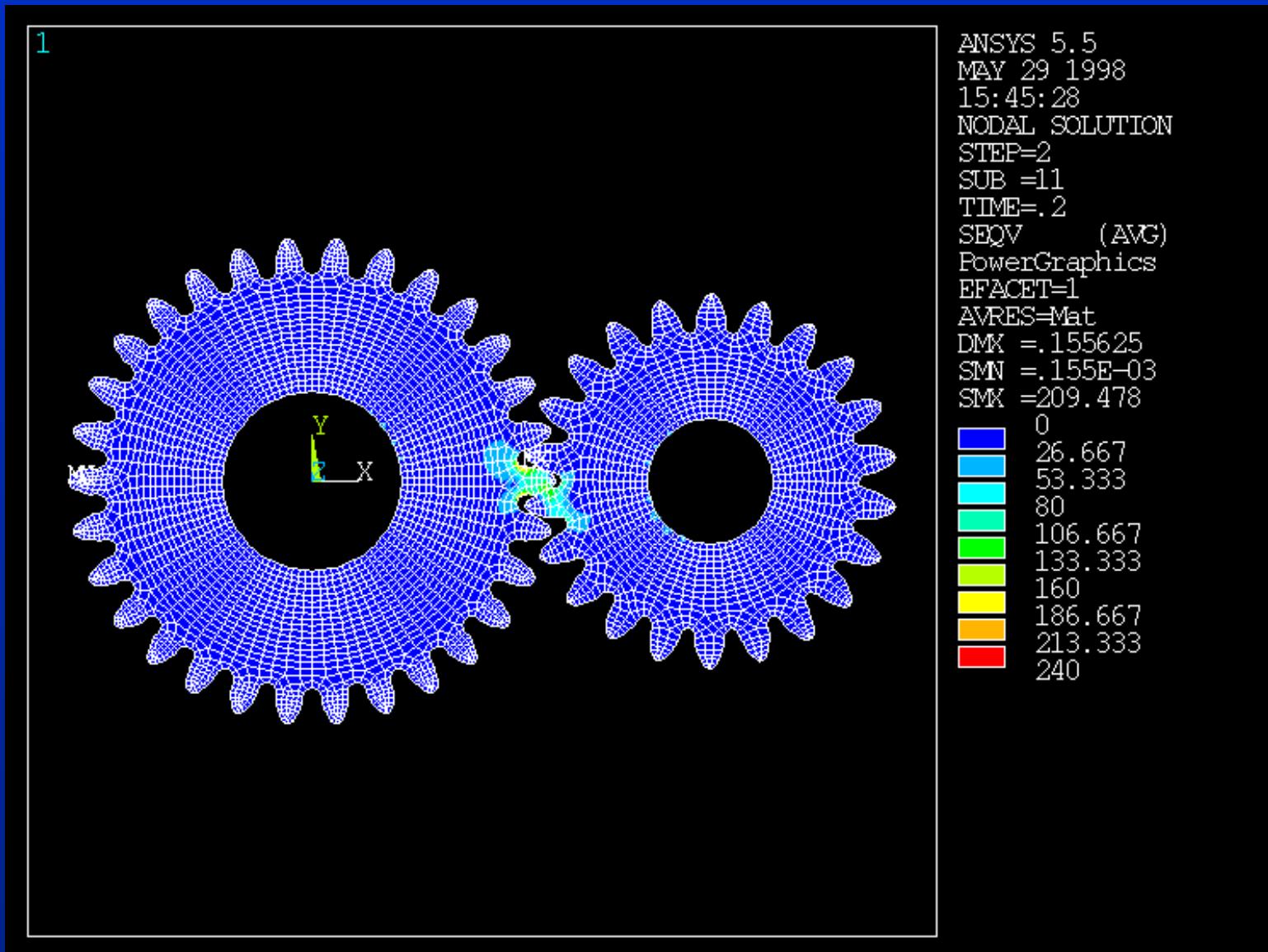
Contact Applications



- Boundary value problems involving contact are of great important in industrial applications in mechanical and civil engineering. The application range includes:
 - Metal forming, drilling problems, bearings, crash analysis, rubber seal, car tires, cooling of electronic devices
 - Biomechanics where human joints, implants or teeth are of consideration.

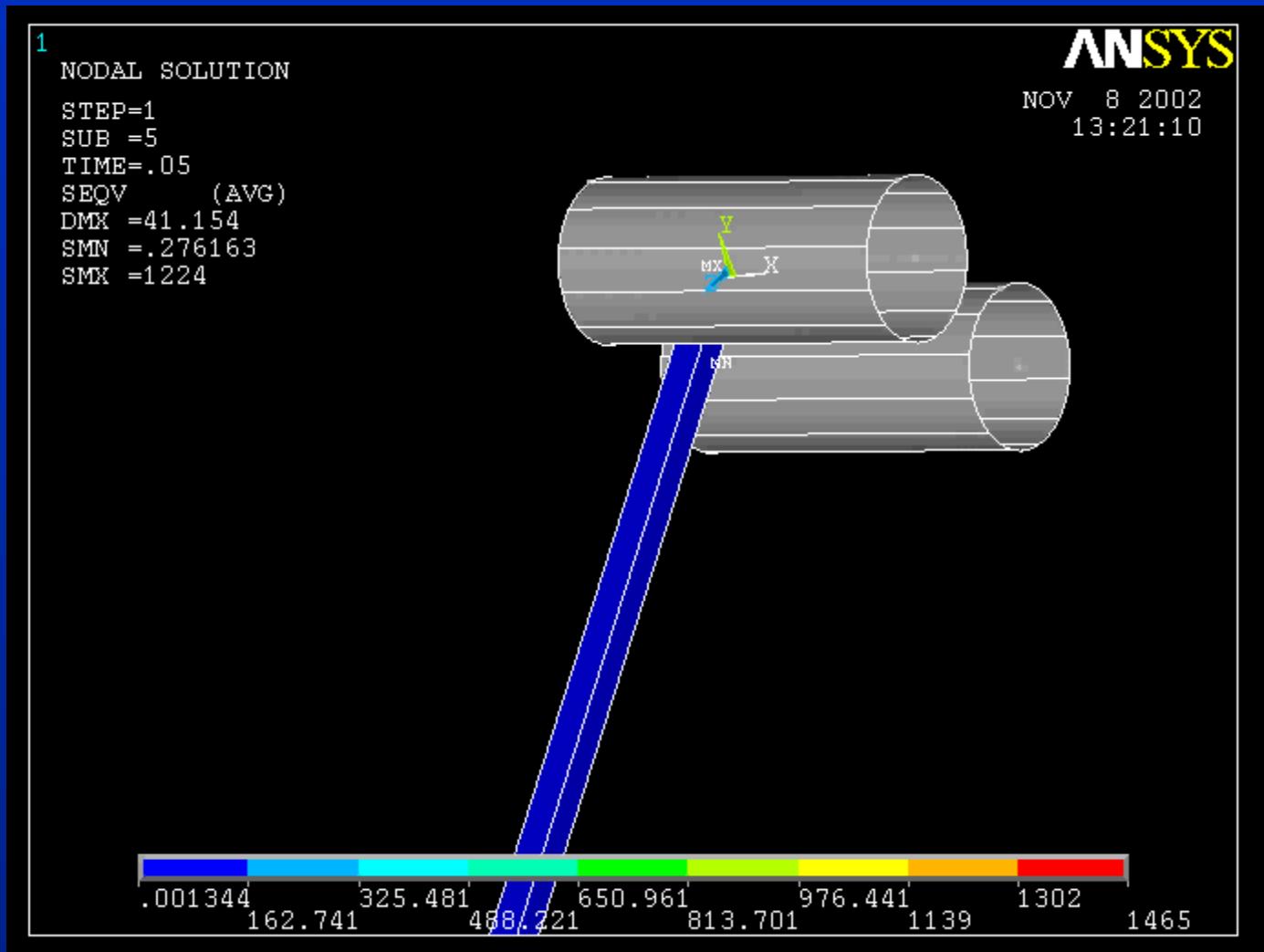
Application: 2D Gear Model

ANSYS



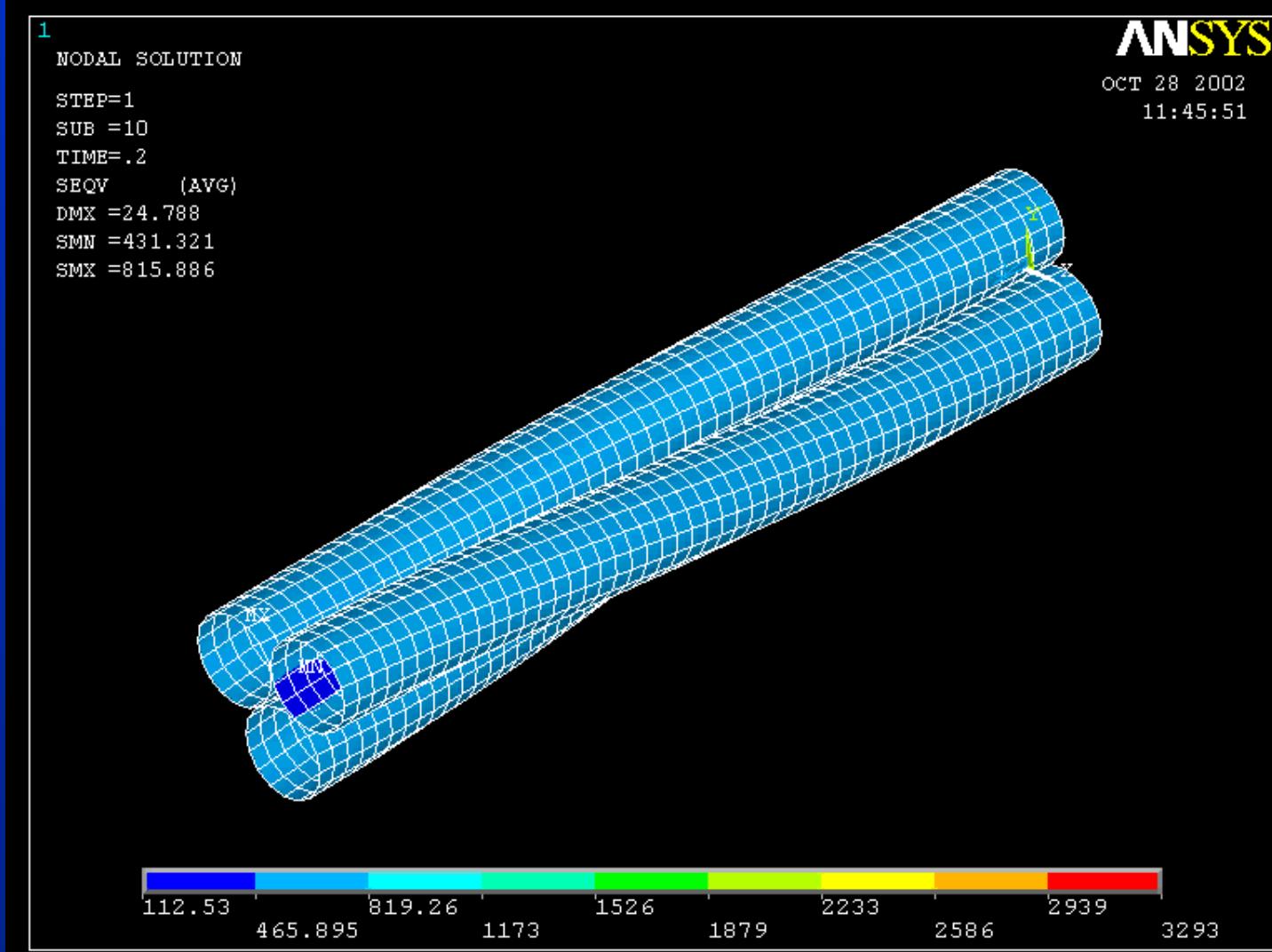
Application: Wiring

ANSYS



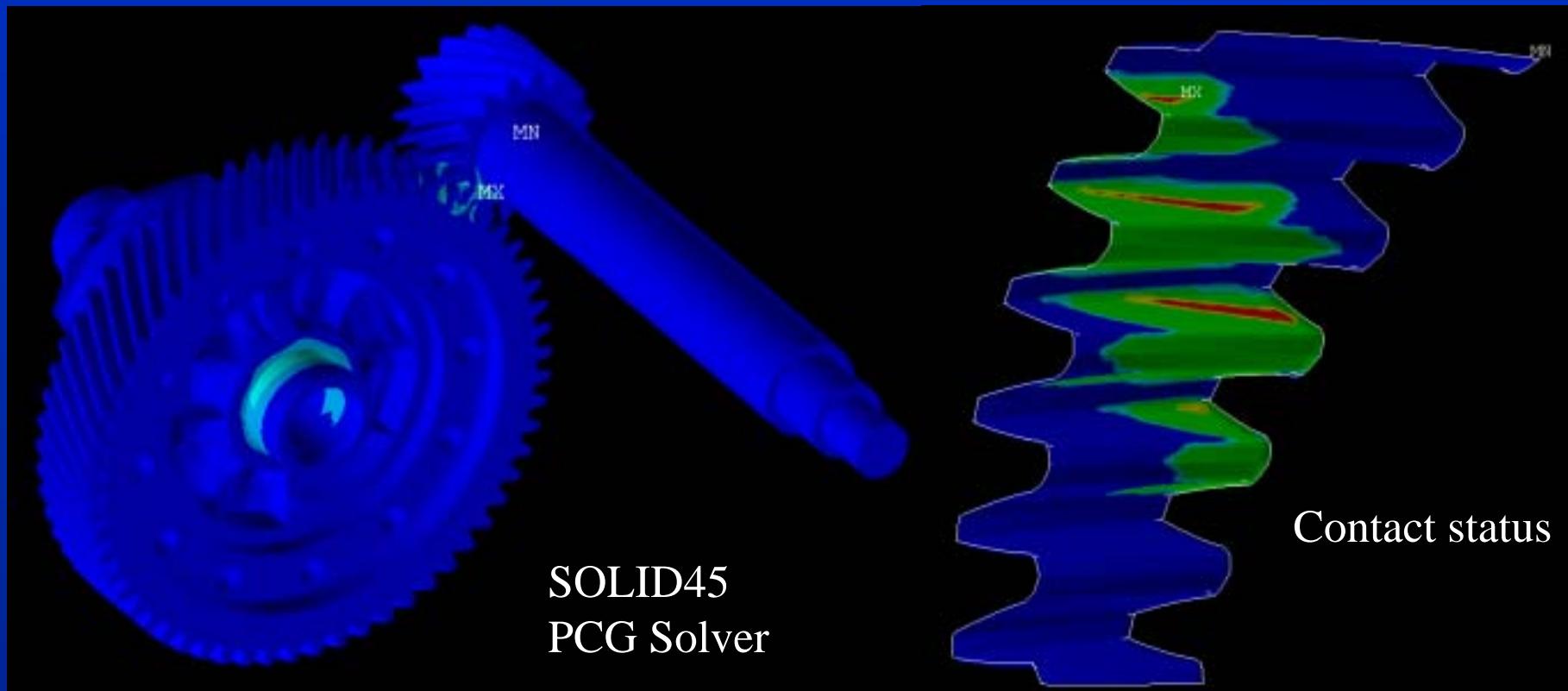
Application: 3D Rope Forming

ANSYS



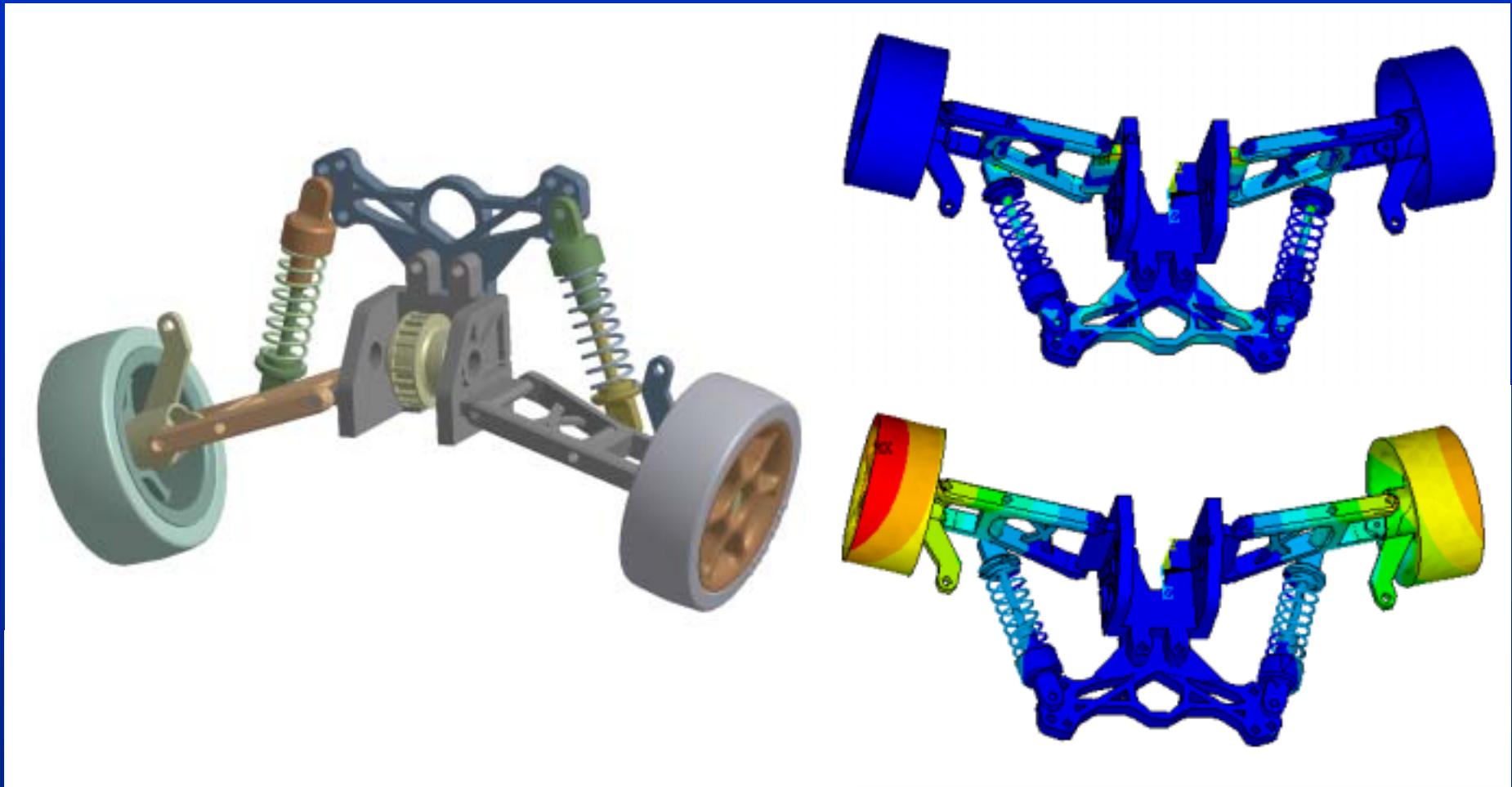
Application: 3D Gear Model

ANSYS



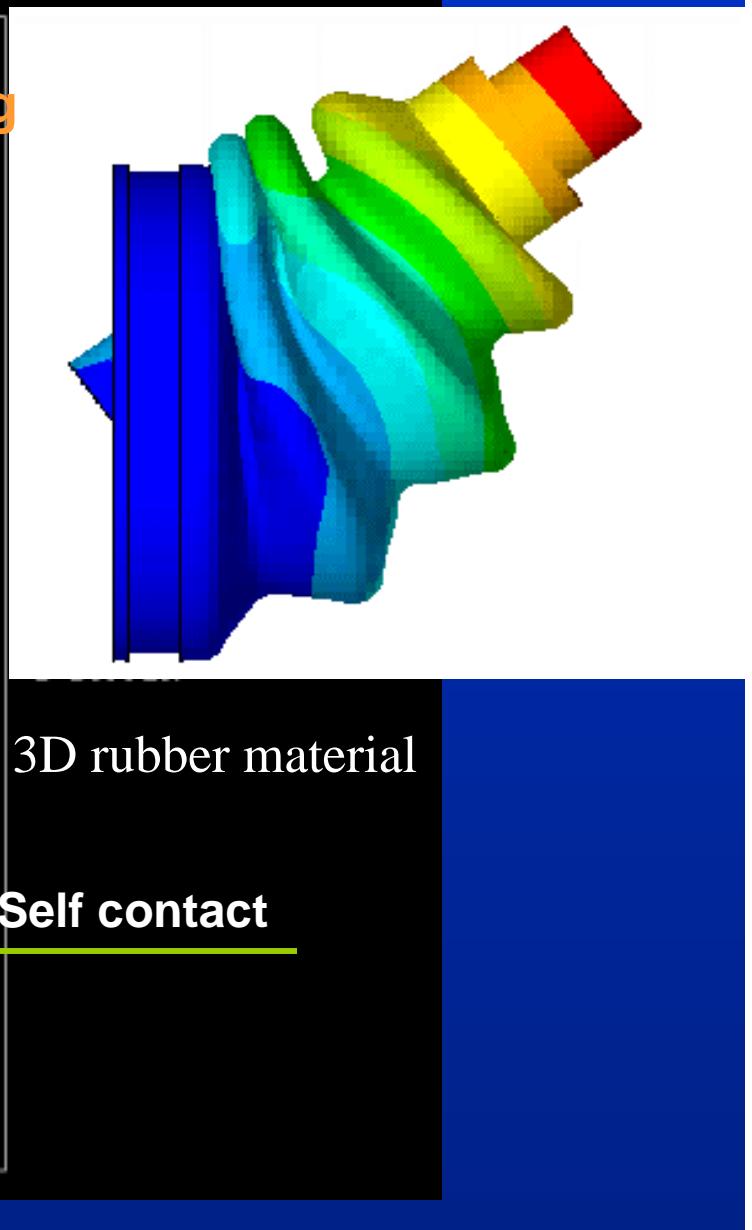
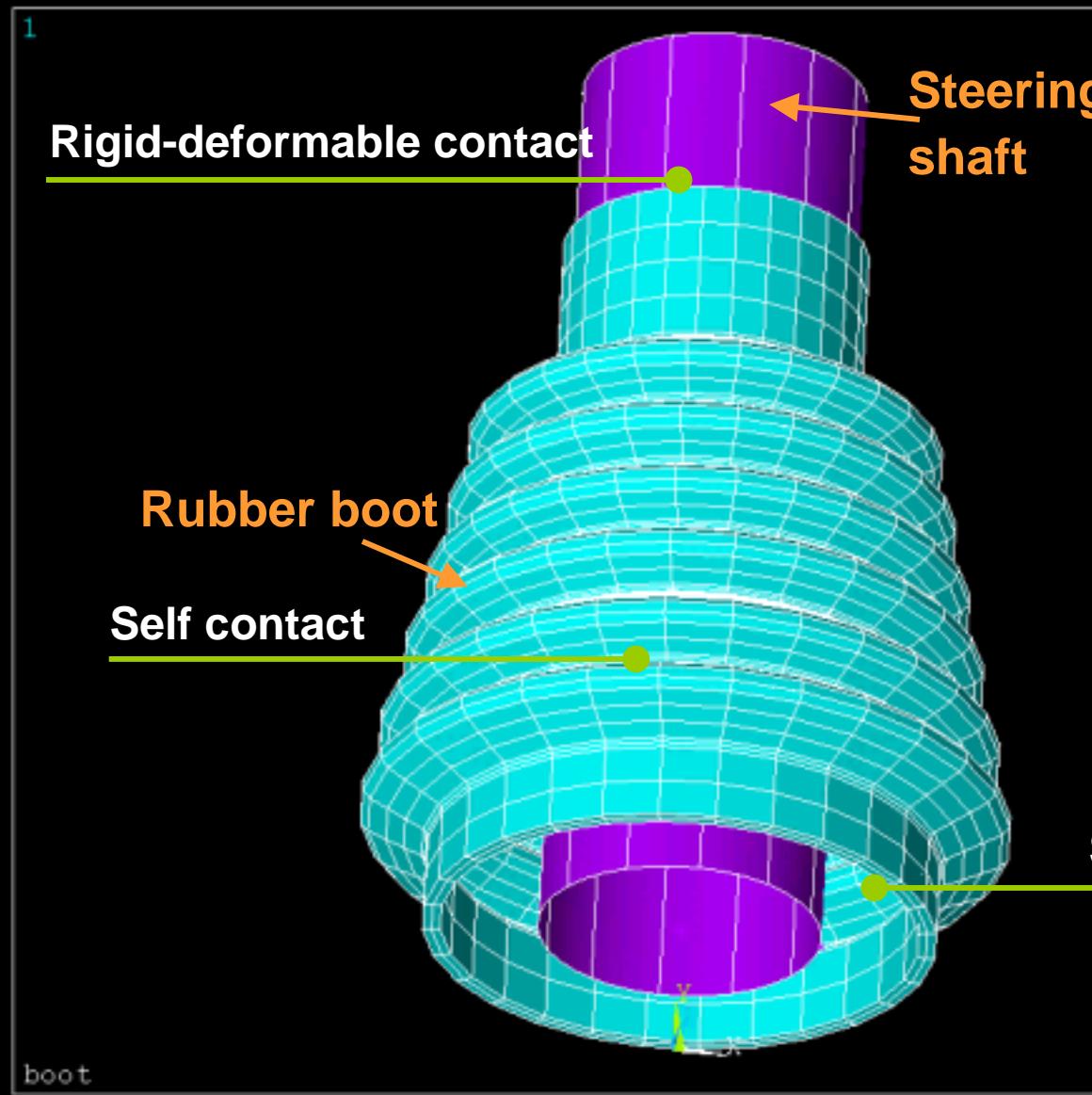
Application: Suspension Model

ANSYS



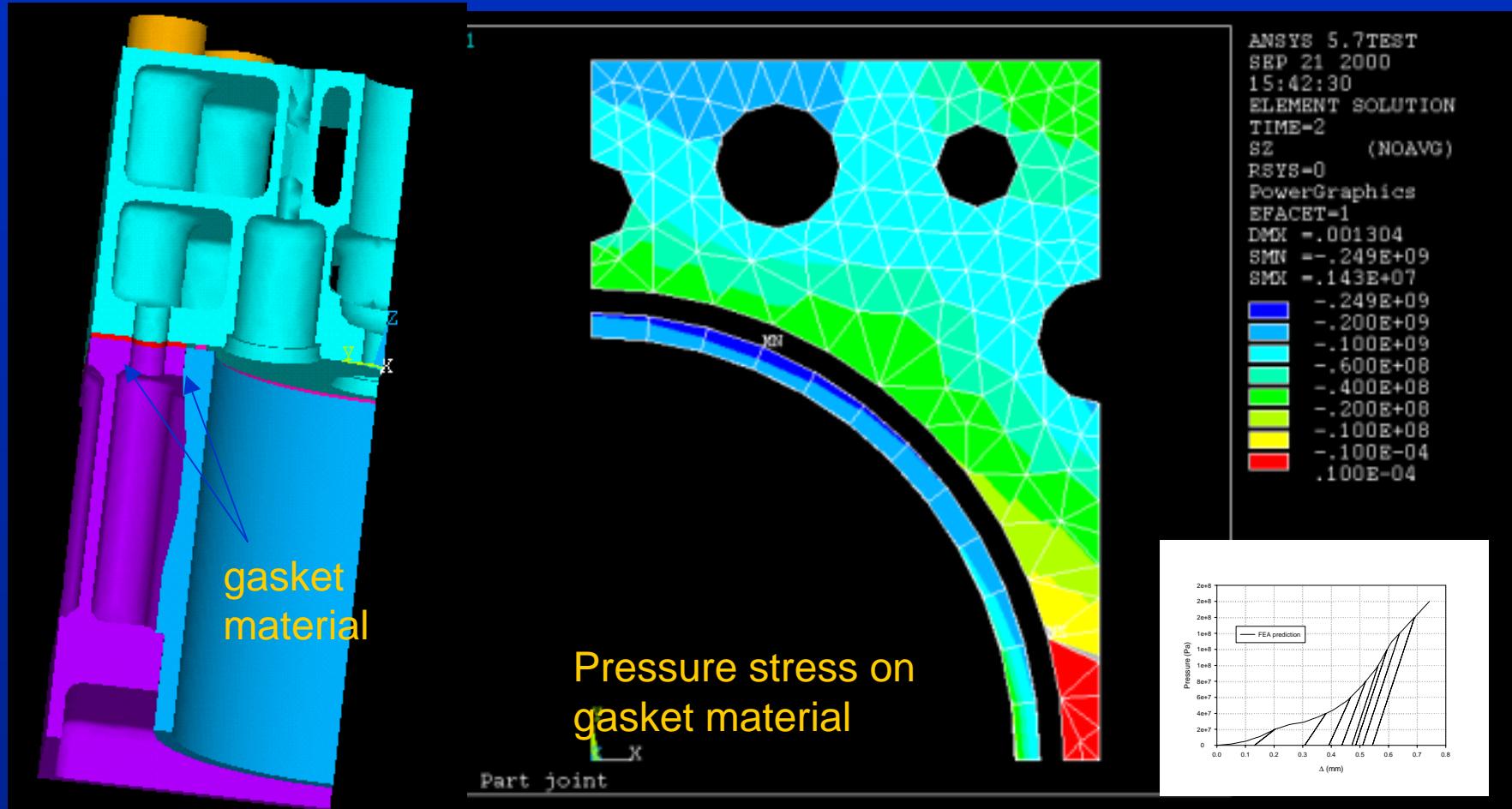
Application: Rubber Boot Seal

ANSYS



Gasket Element/Material Simulation

ANSYS



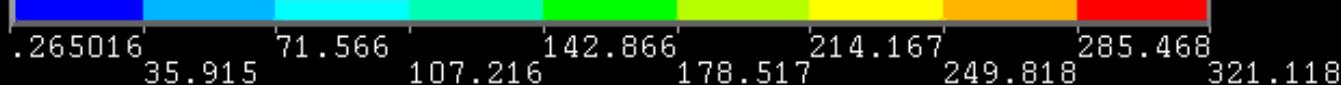
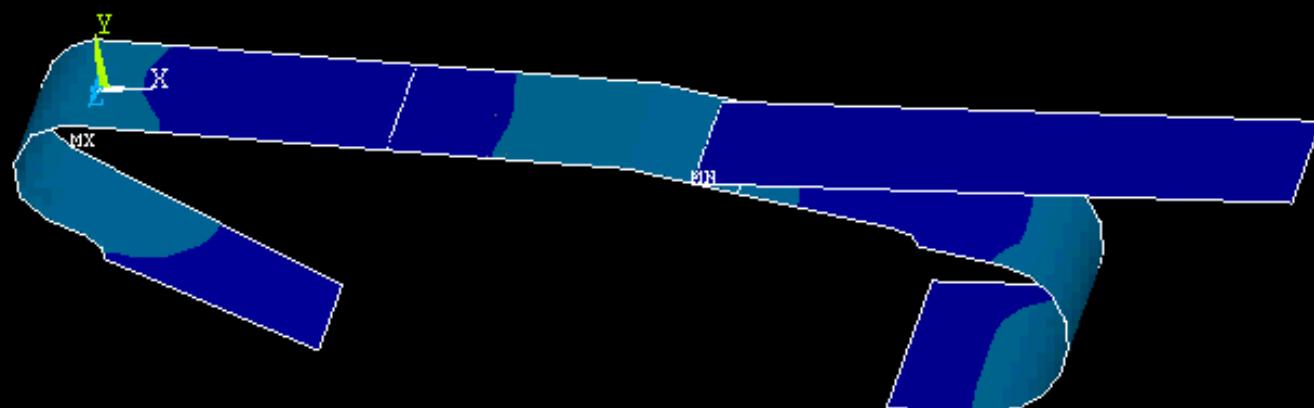
Electrical Connector

ANSYS

```
1  
NODAL SOLUTION  
STEP=1  
SUB =1  
TIME=.05  
SEQV   (AVG)  
DMX  =.15  
SMN  =.265016  
SMX  =74.185
```

ANSYS

FEB 6 2002
16:26:52

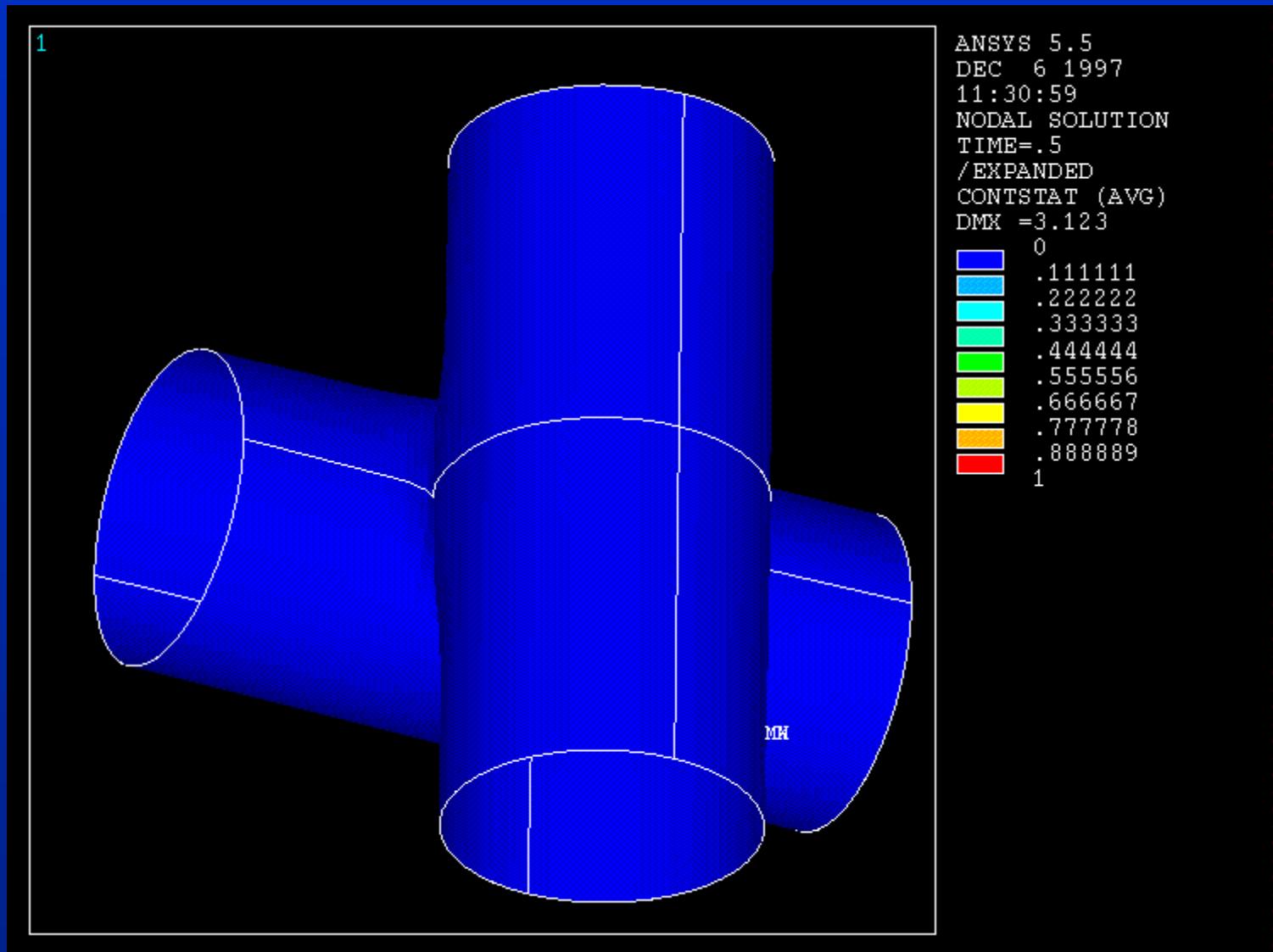


electrical connector

Application: Impact of Two Cylinders

ANSYS

SHELL181
Plasticity
Large strain



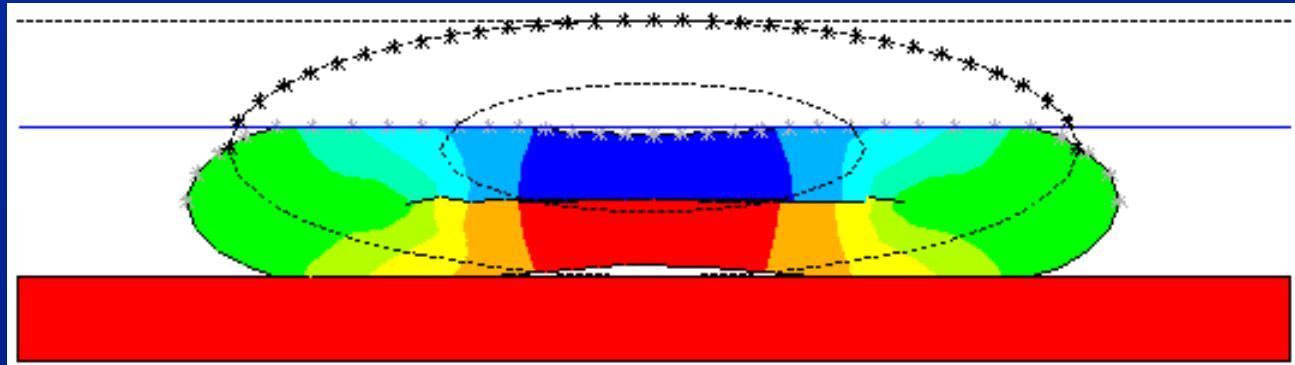
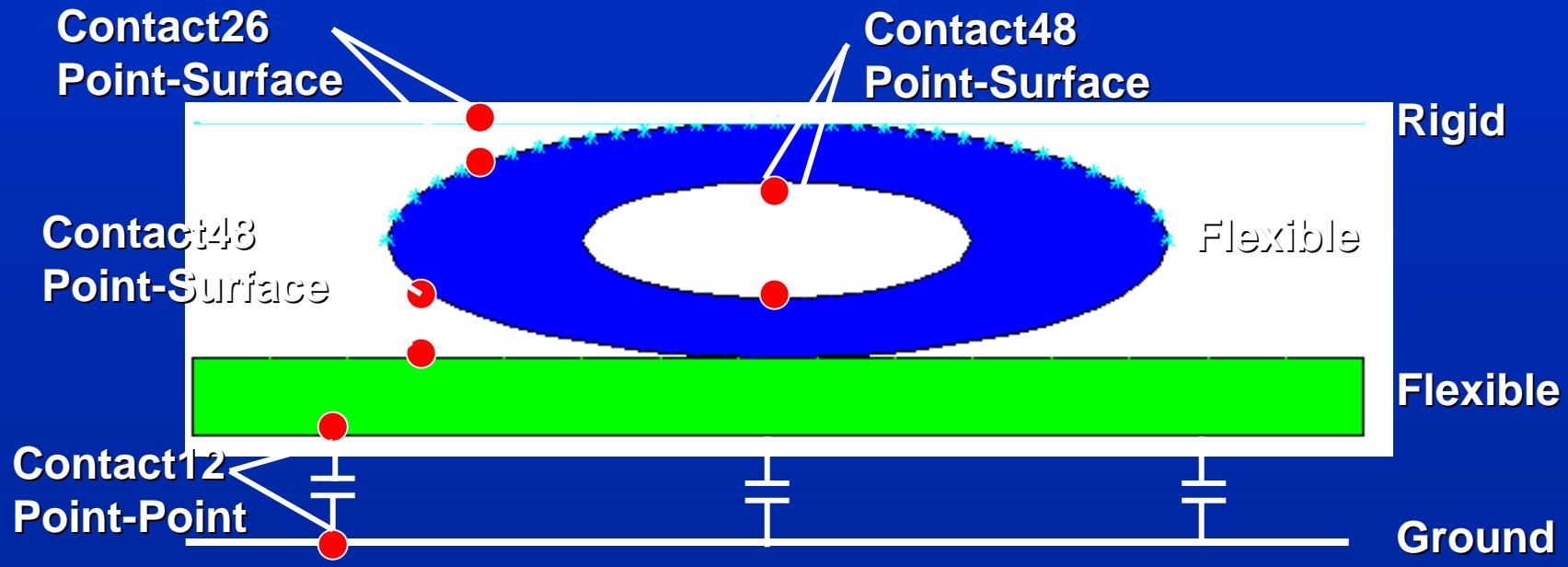
ANSYS Contact Background



	CONTAC12 CONTAC52	CONTA178	CONTAC48 CONTAC49	CONTA175	CONTA171- CONTA174
Type	Node-Node	Node-Node	Node-Surface	Node-Surface	Surf-Surf
Sliding	Small	Small	Large	Large	Large
High order					Yes
Augmented Lagrange		Yes	Yes	Yes	Yes
Pure Lagrange		Yes		Development	Development
Contact stiffness	User defined	Semi-automatic	User defined	Semi-automatic	Semi-automatic
Thermal Electric			Yes	Yes Yes	Yes Yes
Mesh tool	EINTF	EINTF	GCGEN	ESURF	ESURF

ANSYS Contact Background

ANSYS



ANSYS Contact Background



- **CONTA12,52,26,48,49 exist** only because of the historical reason
- These elements are not under continue development
- All the new nonlinear tools, like solution control options are not applied to these elements.
- If you use these elements and at the same time uses **SOLCON,ON**, the solution could be very inefficient.
- If you would like to use elements mentioned above, the following options might help:

SOLCON,OFF

!uses the old nonlinear tools

CNVT,U

!Node-to-Node contact converges much fast with U

MP,MU,1,0

!Friction make the convergence difficult

R,1,1E6

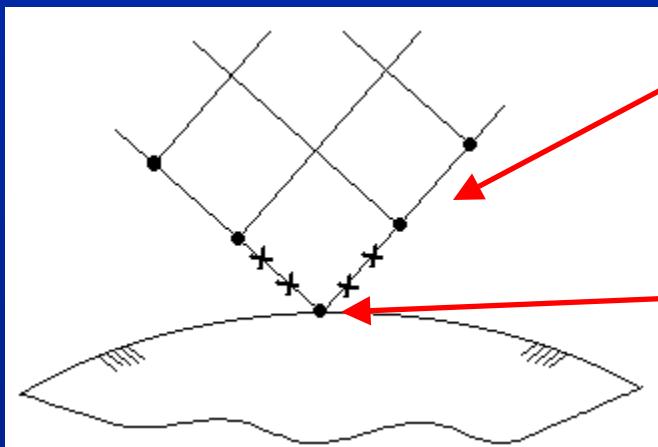
!for E-Modules 2.1e5

For the efficient reason, the above elements should be replaced with

Contact171-175,178, Target169,170

- **Node-node contact element 178**
 - Model point-to-point contact
 - Pipe whip - contact point is always located between the pipe tip and the restraint
- **Node-surface contact element 175**
 - Model point/edge contact
 - Snap-fit - contact can occur around corners
- **Surface-surface contact elements 171-174**
 - Model surface contact
 - interference fit
 - metal forming
- **Mixed contact element types**

- If you have a problem which includes contact at a sharp corner, the surface-to-surface elements, which use the Gauss point as the contact detection point, can experience an over penetration at the corner.
- In such cases, you can mix the surface-to-surface contact elements with node-to-surface elements.



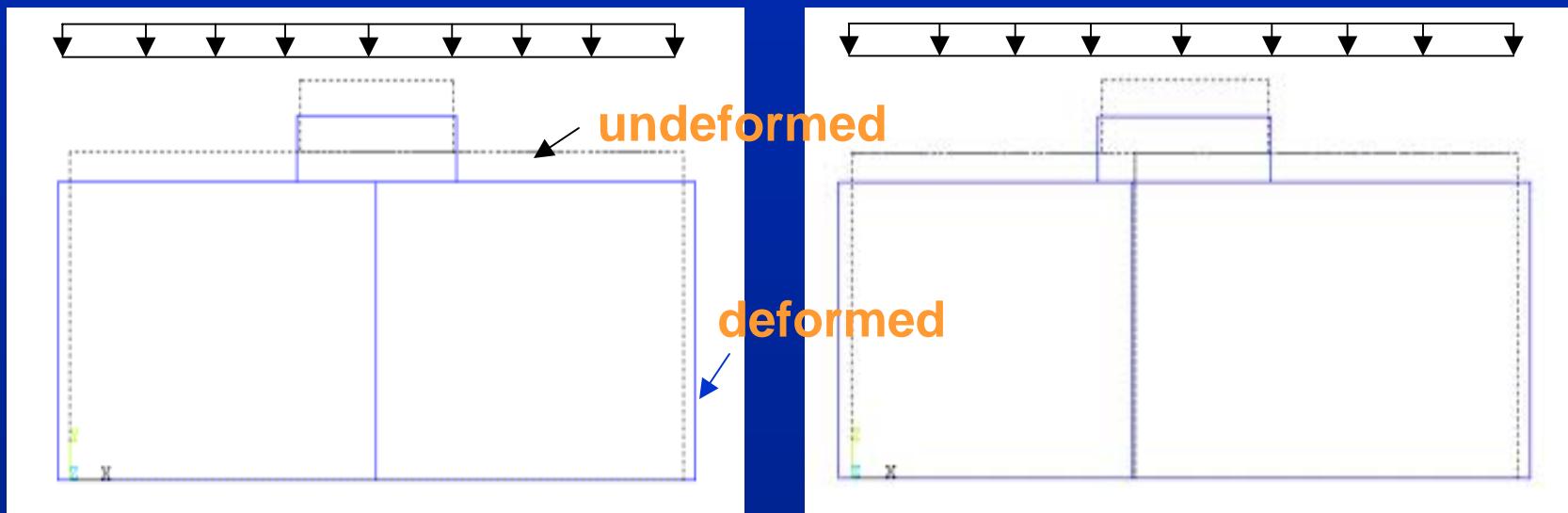
**Surface-to-Surface
Elements**

**Node-to-Surface Elements
used to model the contact
at the corner**

Patch Test

ANSYS

- A uniform pressure is applied on top of surface
- Uniform stress state should be obtained irrespective of the mesh



Patch Test



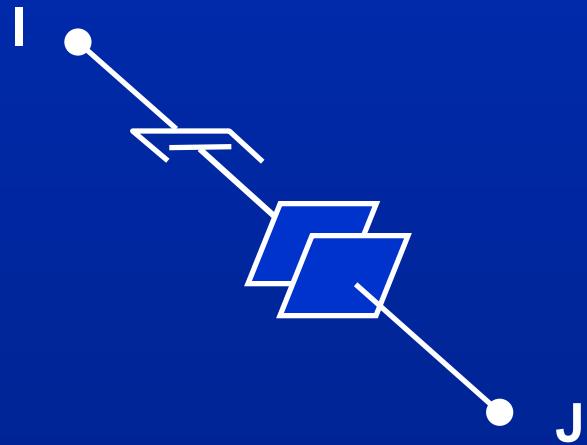
- **Node-node contact pass the patch test**
- **Node-surface contact fails the patch test**
- **Surface-surface contact conditionally pass the patch test**
- **Weak enforcement of contact constrains (penalty method), rather than strong enforcement (Lagrange multiplier method), is the recipe by which patch tests of this type may be passed.**
- **Local penetration violates compatibility condition of displacement based FEA**
 - **Cross penetration into coarsely discretized contact surfaces can lead to inaccurate solution**
 - **Refinement of contact surface leads to global accuracy, although local contact stress oscillation may still be observed**
 - **Only matching meshes completely eliminate these problem**

2D/3D Node-Node Contact Element CONTA178

Node-to-Node Element 178

ANSYS

- It is the simplest and least expensive (in terms of solution CPU) contact element available. When modeling conditions warrant their use, it can be an effective tool for modeling a variety of contact situations.



CONTA178 - 3D Gap (with damping)

Node-node CONTA178: Objectives



- Replacement of old gap elements 12,52,40
- Alternative contact algorithms
- Exact compatibility of contact constraint
- More functionality
- Easy of use and intelligent default settings
 - Chattering & penetration control, contact stiffness
- More efficient and more stable than
 - Node-surf, surf-surf contact elements
- Request from ANSYS critical customers

Limitations of Old Gap Elements



- **CONTAC52**
 - Contact normal based on initial locations of two nodes
 - two nodes must be separated initially
 - may provides wrong contact normal
 - Issues with real geometry model
 - two node are initially coincident or overlap
 - initial interference fit where initial penetration is not constant
 - No penetration control
 - contact stiffness must be an input
 - Lack of cylindrical gap

Limitations of Old Gap Elements



- **CONTAC12**
 - **Contact normal based on real constant**
 - each contact element may have its own real constant set
 - **Circular gap**
 - 2D frictionless
 - **element stiffness is not updated**
- **COMBIN40**
 - **Gap normal based on nodal coordinate**
 - only support one dimension
 - accelerations operate, mass and inertia relief calculations are not correct
 - **Not a true contact element**

Node-Node CONTA178: Overview

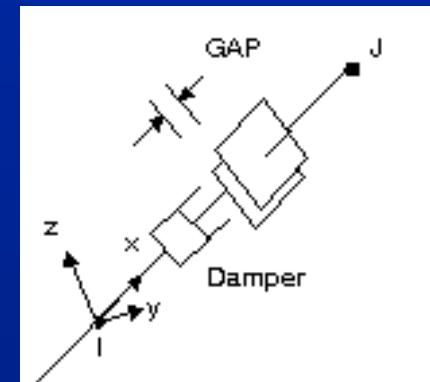


- The contact nodal forces are assumed to be directly conjugate to the nodal gaps.
- The approach can not support 3D higher order element contact
- The approaches is capable of passing so-called contact patch tests and LBB condition due to matching mesh pattern.

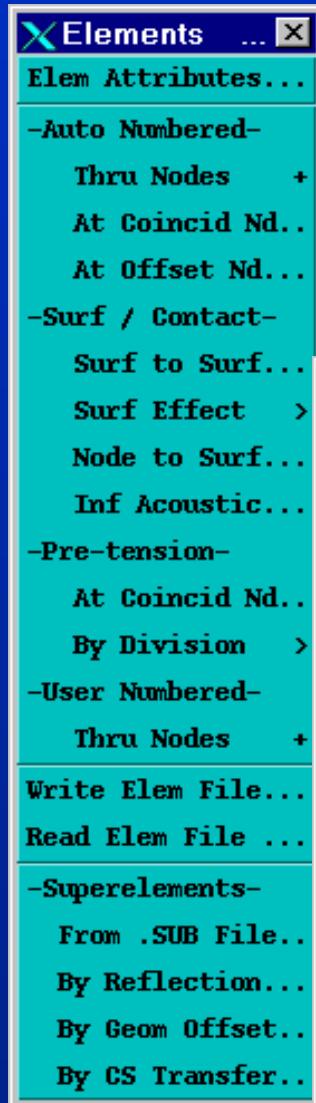
Node-Node CONT175: Overview



- Pure Lagrange, augmented Lagrange, or pure penalty
- algorithms
 - Pure Lagrange permits tiny penetrations (best accuracy)
- User-definable contact normal direction (several options)
- Advanced surface behavior options (bonded, no-separation, ramped interference, etc.)
- User-definable initial gap or interference
- Element damper available (for closed gap status)
- Semi-automatic real constants (factors)
- Frictional circular & cylindrical gap types
- Weak spring option for open gap
- Unsymmetric solver option (NROPT)

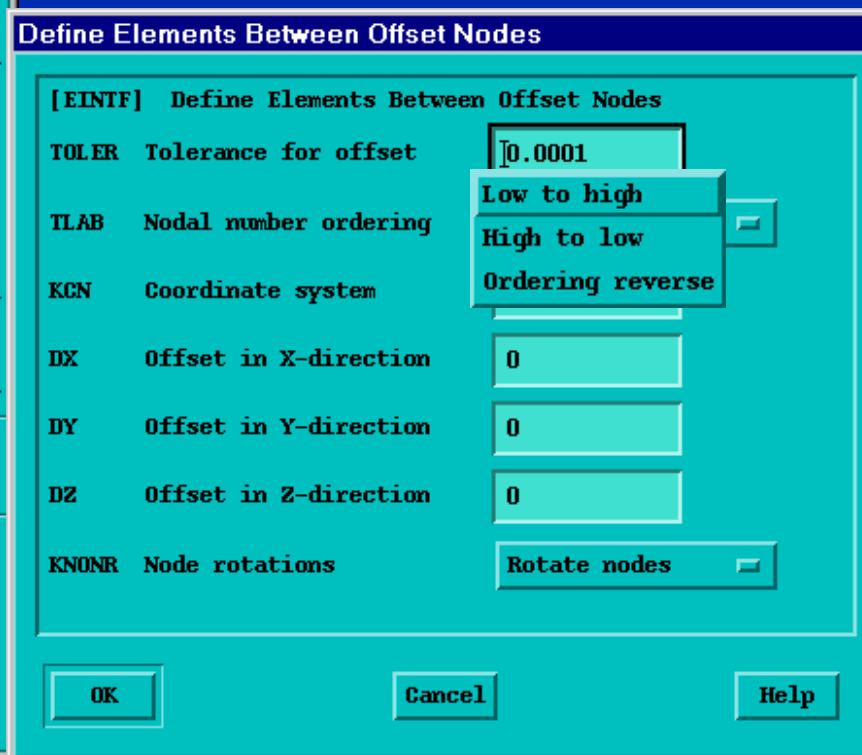


Contact Element Generation



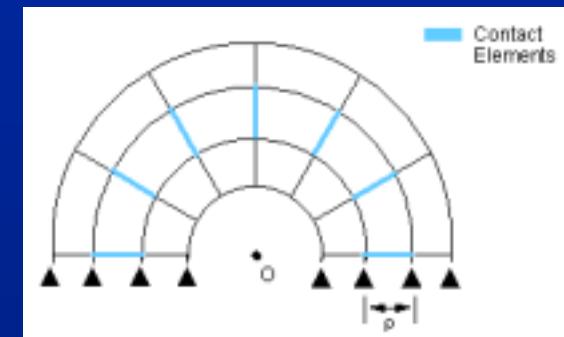
– EINTF enhancements

- User-specified DX, DY, DZ offset values
 - Useful for either coincident or offset (noncoincident) nodes



- User control over node number ordering

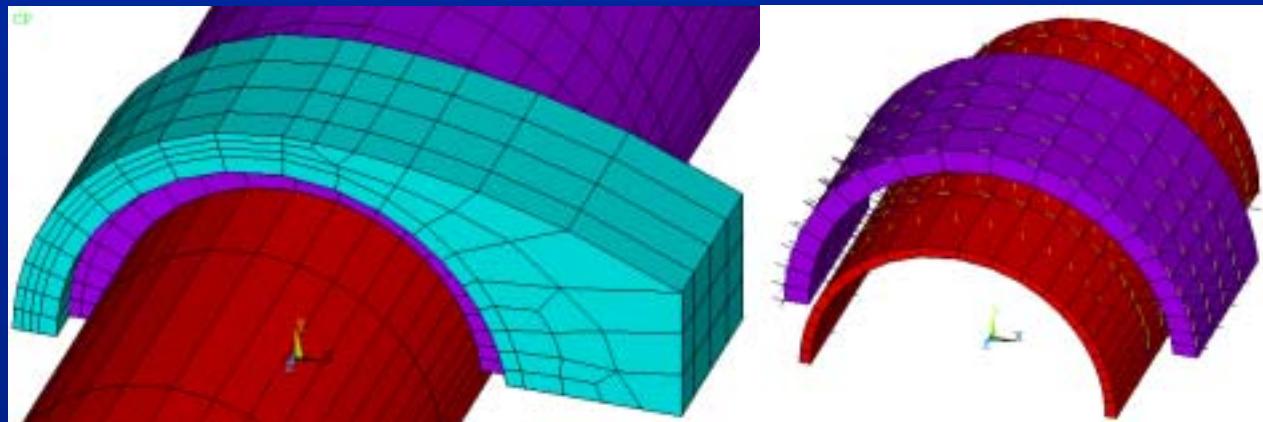
- EINTF,,,REVE flips normal direction



Node Ordering

ANSYS

- Node ordering is critical for contact normal
 - First node on one side, second on other side
- Ordering control: EINTF,,,low/high
- Order display: /PSYM, esys
- Order verification
 - NSEL,s,pos,1 + ESLN + NSLE + /PSYS,esys + EPLOT
- Ordering correction: EINTF,,,reve



Contact Normal

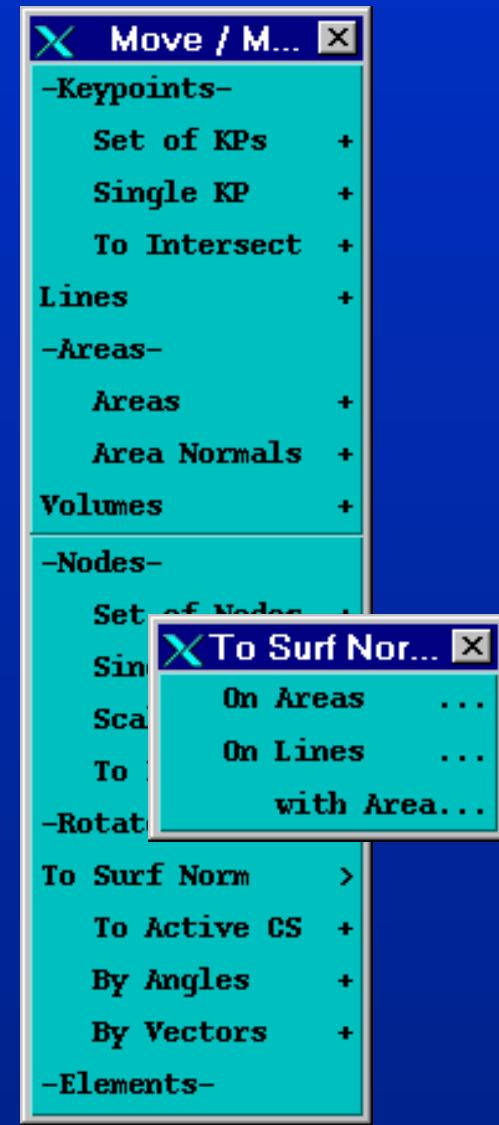


- **Support real geometry model**
 - Nodes may be coincident or overlap
- **KEYOP(5)=0**
 - Real constants (NX,NY,NZ) if defined
 - Node locations (initially separate position)
- **KEYOP(5)=4,5,6**
 - X (Y,Z) axis of element coor. sys.
 - ESYS must be a Cartesian system
 - Each element can has its own

Contact Normal

ANSYS

- KEYOP(5)=1,2,3
 - X (Y,Z) axis of nodal coor. sys.
 - averaging on both nodes
 - Each element has its own
 - Easy to build if using solid model command
 - NORA,area,ndir
 - NORL,line,area,ndir
 - Acceleration, mass and inertia relief are correct



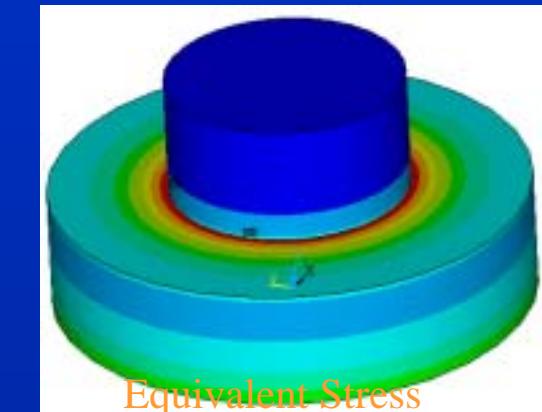
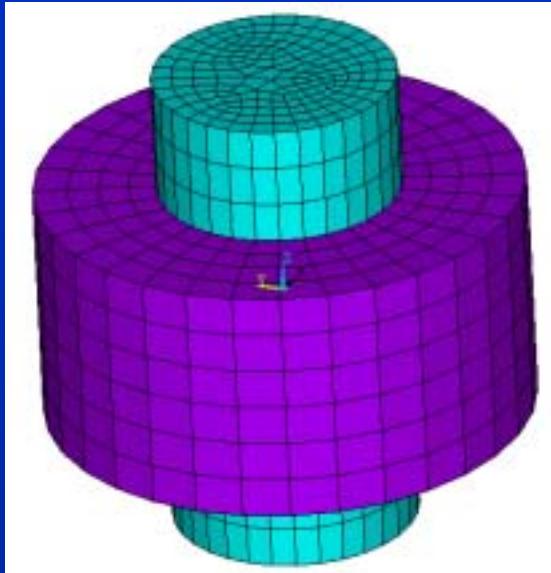
Alternative contact algorithms



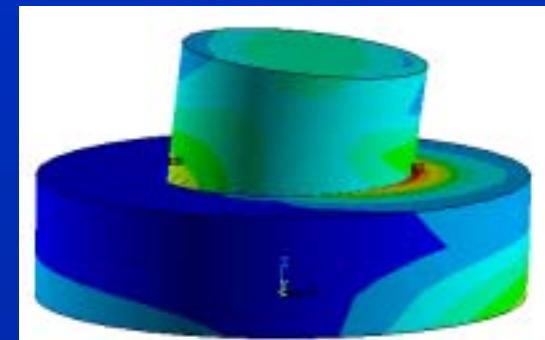
- **Pure Lagrange multipliers method**
 - Near zero penetration and slip, no contact stiffness
 - More DOF, over constraint, chattering
 - Solver issue: PCG, impact, eigenvalue buckling
- **Augmented Lagrange method**
 - Less expensive, more robust
 - Contact elements superposition
 - Less accurate, ill condition if too big contact stiffness
- **Lagrange multiplier on normal & penalty on tangent**
 - Behaviors between above two

3D Contact Model

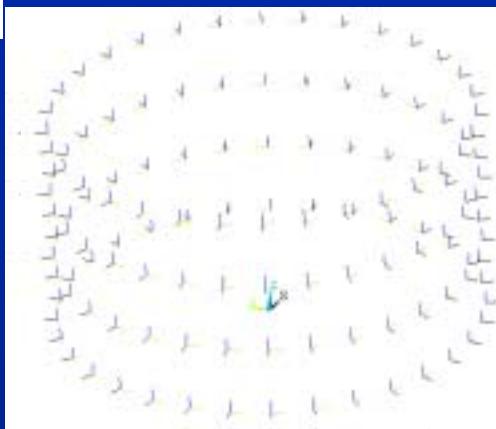
ANSYS



Equivalent Stress
after interference fit



Equivalent Stress
after bending



Contact normal



Contact status

Alternative contact algorithms



	Lagrange unsymmetry	Lagrange symmetry	Lagrange Penalty	Augmented Lagrange
DOF	5722	5722	5466	5338
Iteration	15	17	13	14
Penetration	<1.e-15	<1.e-7	<1.e-14	<1.e-4
CPU	322	278	221	230

Alternative contact algorithms



- Pure Lagrange multiplies method
 - KEYOP(2)=0
 - Contact length: average distance from contact node to center of underlying element
 - Chattering control parameters
 - TOLN: Max. overlap contact remains open
 - 0.1*Max. Displacement convergence tolerance
 - 1.e-3*contact length
 - FTOL: Max. tension force contact remains close
 - 0.1*Max. residual force

Lagrange Multiplier Method



```
17 CONTACT POINTS HAVE CHANGE OF CONTACT STATUS
FORCE CONVERGENCE VALUE = 5691.      CRITERION= 130.0
DISP CONVERGENCE VALUE = 4.802      CRITERION= 0.3211E-01
EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.8028E-01
LINE SEARCH PARAMETER = 0.9947      SCALED MAX DOF INC = -0.7985E-01
74 CONTACT POINTS HAVE CHANGE OF CONTACT STATUS
56 CONTACT POINTS VIOLATE COMPATIBILITY OF CONTACT CONSTRAINT
FORCE CONVERGENCE VALUE = 0.1586E+05 CRITERION= 207.2
DISP CONVERGENCE VALUE = 0.6195      CRITERION= 0.3211E-01
EQUIL ITER 2 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1171E-01
LINE SEARCH PARAMETER = 1.000       SCALED MAX DOF INC = 0.1171E-01
6 CONTACT POINTS HAVE CHANGE OF CONTACT STATUS
5 CONTACT POINTS VIOLATE COMPATIBILITY OF CONTACT CONSTRAINT
FORCE CONVERGENCE VALUE = 1024.      CRITERION= 206.3
DISP CONVERGENCE VALUE = 0.3157E-01 CRITERION= 0.3211E-01 <<< CONVERGED
EQUIL ITER 3 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.1179E-02
LINE SEARCH PARAMETER = 0.9869      SCALED MAX DOF INC = -0.1164E-02
1 CONTACT POINTS HAVE CHANGE OF CONTACT STATUS
FORCE CONVERGENCE VALUE = 380.8      CRITERION= 206.3
DISP CONVERGENCE VALUE = 0.9317E-02 CRITERION= 0.3211E-01 <<< CONVERGED
EQUIL ITER 4 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.4712E-03
LINE SEARCH PARAMETER = 1.000       SCALED MAX DOF INC = -0.4712E-03
FORCE CONVERGENCE VALUE = 10.16      CRITERION= 206.3      <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 4
*** LOAD STEP 2 SUBSTEP 1 COMPLETED. CUM ITER = 8
*** TIME = 1.50000    TIME INC = 0.500000
*** AUTO STEP TIME: NEXT TIME INC = 0.50000    UNCHANGED
```

Contact status
adjustment and
Compatibility check

Alternative contact algorithms



- Lagrange multiplier on normal and Penalty on frictional plane
 - KEYOP(2)=1
 - Max. slip control SLTOL
 - 5.e-3*contact length
 - used to determine tangent contact stiffness KS
 - $KS = \max(\mu * f_n / SLTOL, rvr(4))$
 - Chattering control parameters
 - TOLN, FTOL

Alternative contact algorithms



- Augmented Lagrange method
 - KEYOP(2)=2
 - Normal contact stiffness KN
 - Contact length*Elastic Modulus (EX) of underlying elements
(contact length*1.e9 if EX=0)
 - Penetration tolerance TOLN
 - 5.e-3*contact length
 - degenerate to penalty method if using big TOLN
 - Max. slip control SLTOL and KS
- Pure Penalty method
 - KEYOP(2)=3
 - Contact stiffness KN, KS

Initial Gap size

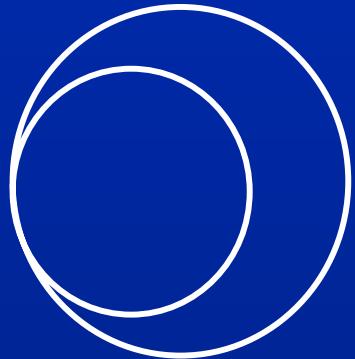


- Real constant GAP + initial node locations
(KEYOPT(4)=0)
 - Depends on contact normal
 - Not a distance
 - Supports varying initial interference
- Real constant GAP (KEYOP(4)=1)

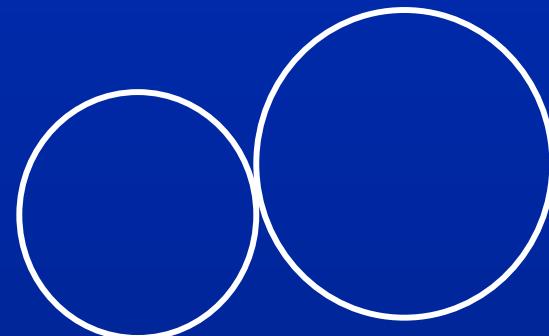
Cylindrical Gap

ANSYS

- Set KEYOPT(1)=1 & ignore KEYOPT(4,5)
- Cylindrical axis direction cosines
 - Real constant NX, NY, NZ. (0,0,1 as default)
- Cases



Case 1: $GAP = r_2 - r_1 > 0$
 $un = GAP - |x_2 - x_1| \geq 0$



Case 2: $GAP = -(r_1 + r_2) < 0$
 $un = |x_2 - x_1| + GAP \geq 0$

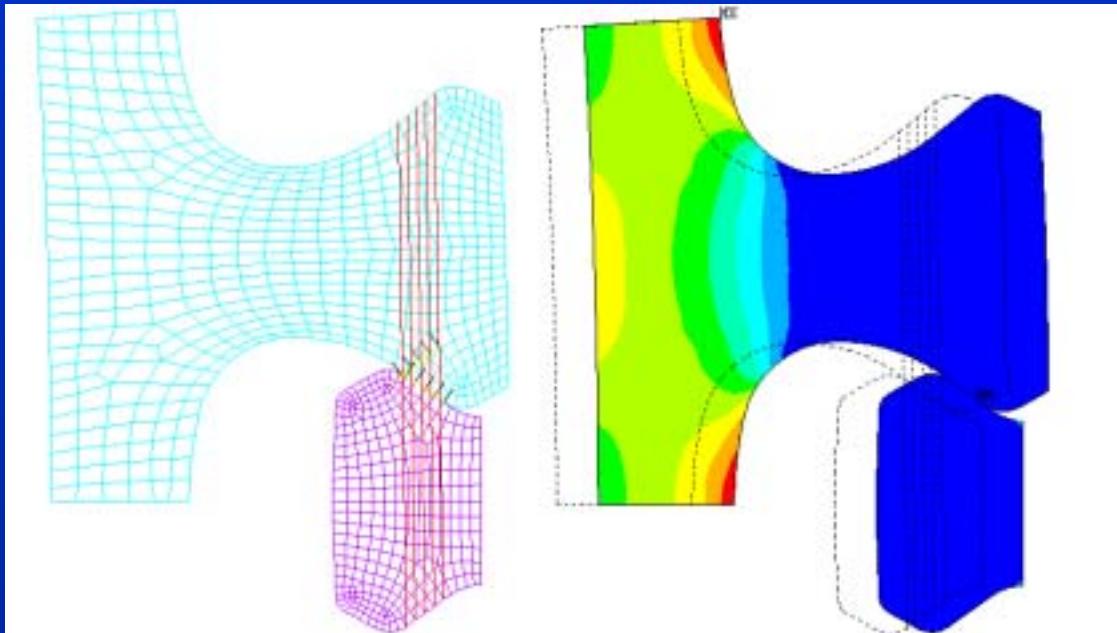
Weak Spring Options



- KEYOP(3)=0 does not use
- KEYOP(3)=1 acts across an open gap
- KEYOP(3)=2 open gap and/or sliding plane
 - Only contributes to contact stiffness
- KEYOP(3)=3 acts across an open gap
- KEYOP(3)=4 open gap and/or sliding plane
 - Contributes to contact stiffness and force
- Real constant REDFACT (default=1.e-6)
- Do not combine with no-separate & bonded

Application: 2D Cyclic Sector Contact

ANSYS



Contact normal based on
nodal coordinate system

Equivalent stress
after spin

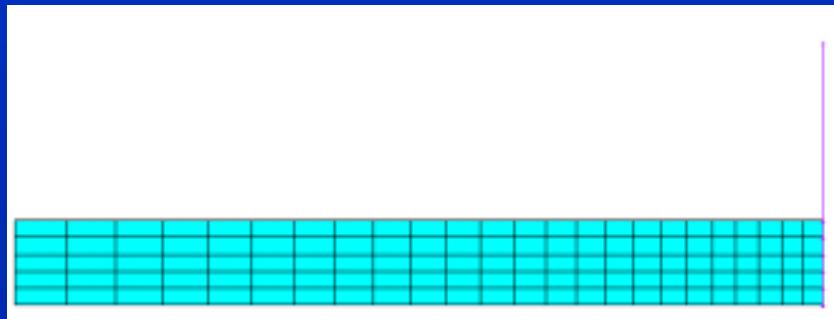
Contact results

XPRETAB Command

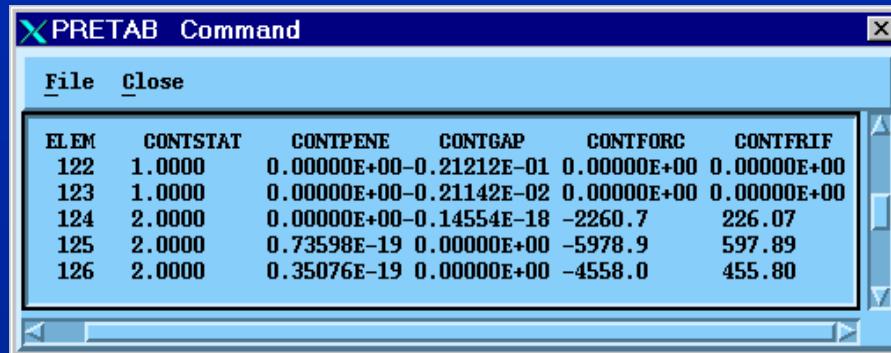
ELEM	CONTSTAT	CONTIPENE	CONTGAP	CONTFORC	CONTRIF
804	2.0000	0.54295E-09	0.00000E+00	-152.90	45.871
805	2.0000	0.41914E-09	0.00000E+00	-154.17	46.251
806	2.0000	0.62816E-09	0.00000E+00	-175.91	52.774
807	2.0000	0.29277E-09	0.00000E+00	-204.78	61.433
808	2.0000	0.20165E-09	0.00000E+00	-300.85	90.255
810	3.0000	0.00000E+00	-0.15074E-16	-269.62	-70.166
811	3.0000	0.18105E-17	0.00000E+00	-189.52	-37.049
812	2.0000	0.00000E+00	-0.79434E-09	-155.35	-46.606
813	2.0000	0.00000E+00	-0.48723E-09	-170.89	-51.268
814	2.0000	0.00000E+00	-0.44923E-09	-212.40	-63.721

Application: Bar Impact on a Rigid Wall

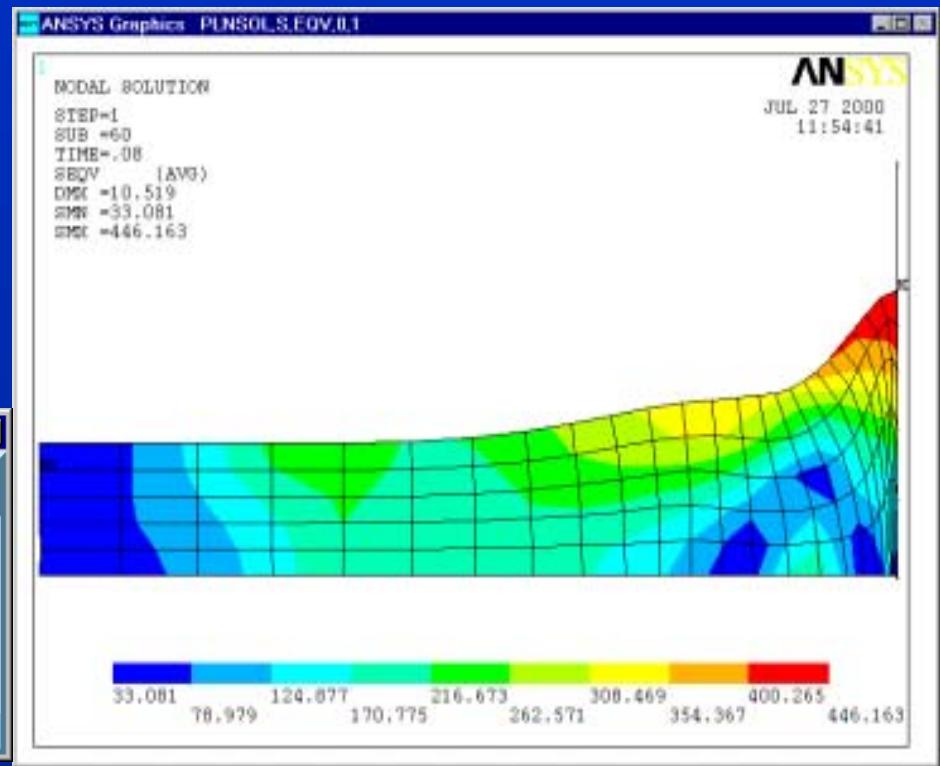
ANSYS



Initial shape



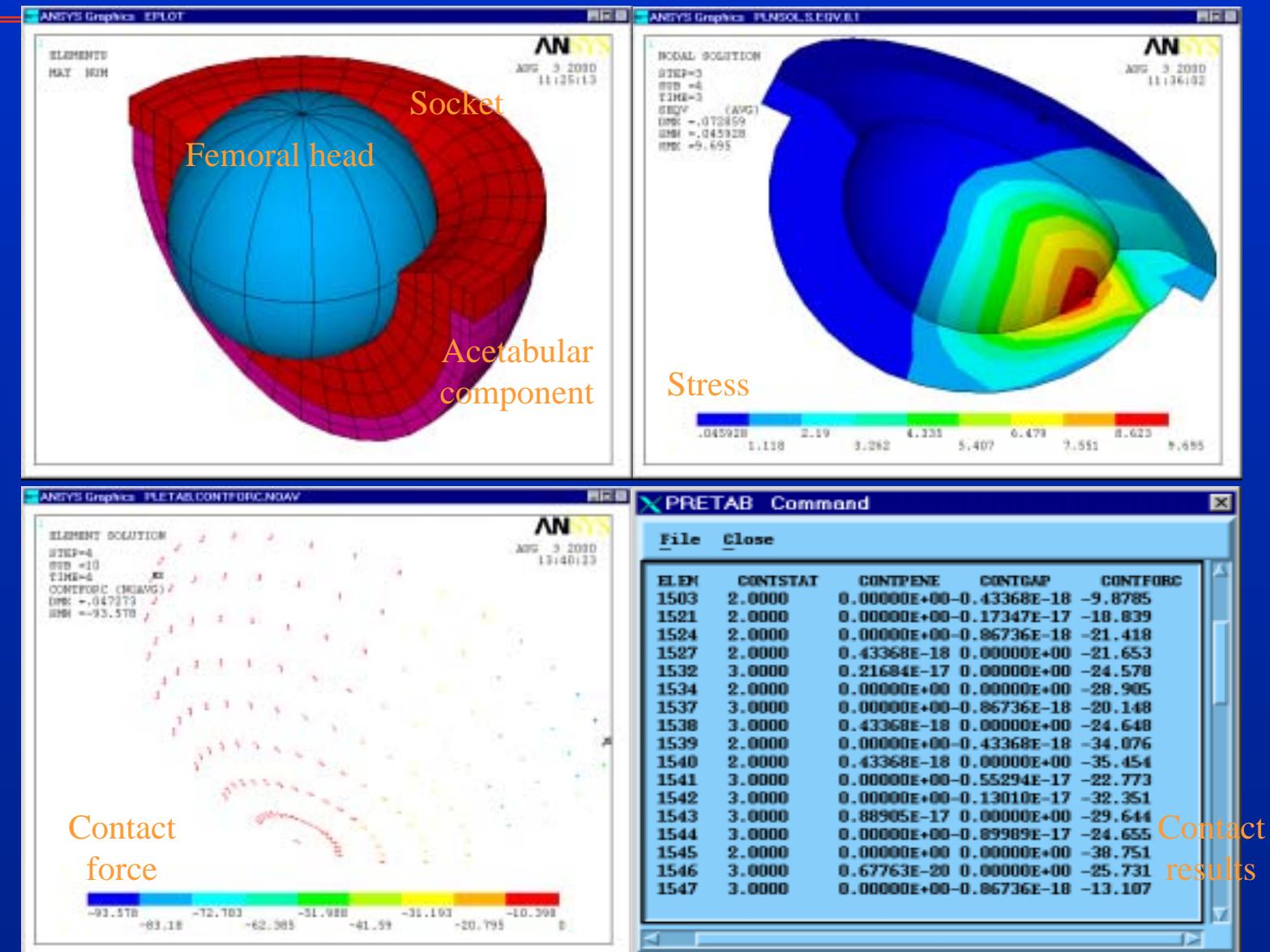
Contact element results



Equivalent stress

Application: Artificial Hip Joint

ANSYS



Node-Node CONTA178: Limitation

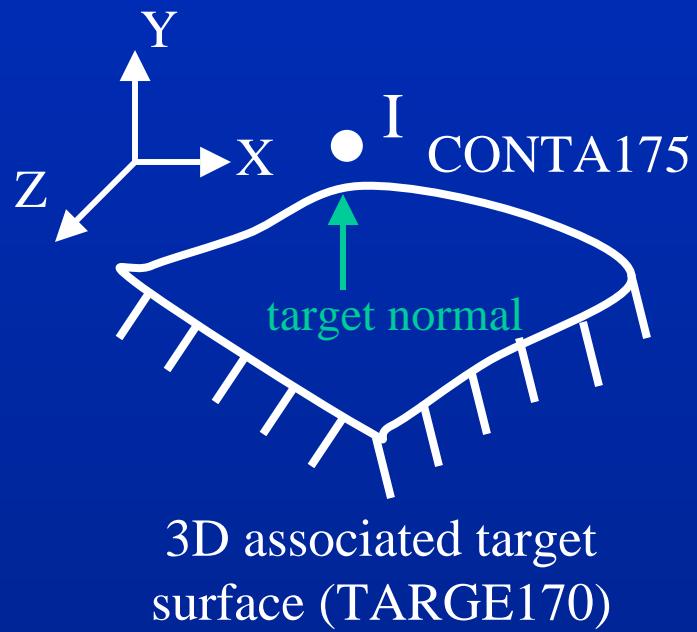
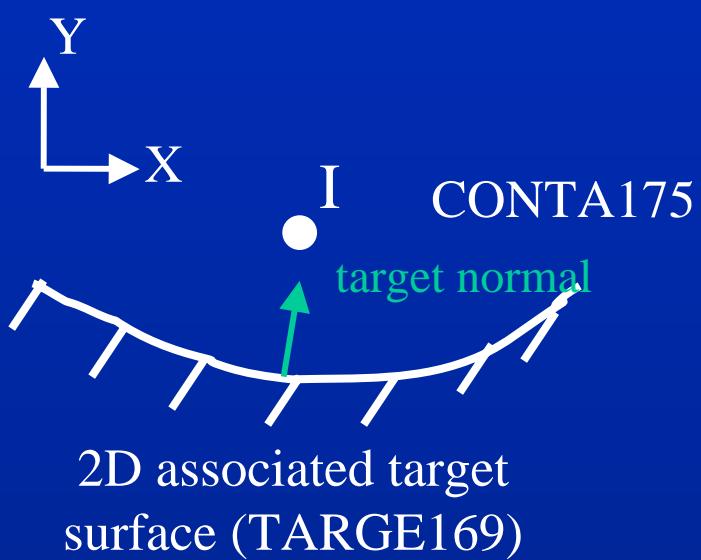


- Requires matching mesh pattern on both side of contact face
- Requires 8 nodes hexahedron
- Only supports small sliding and rotation
- Does not support multi-physics contact
- Contact results can not be visualized

2D/3D Node-Surface Contact Element CONTA175

Node-Surface Contact Element 175

ANSYS



Target Segments



Line



Parabola



Arc



Circle

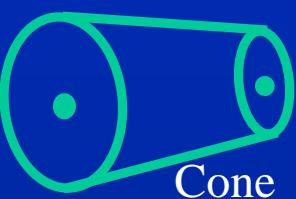


Pilot node

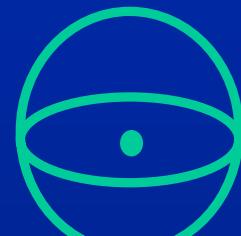
TARGE169



Cylinder



Cone



Sphere



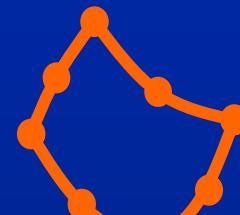
Tri



Tri6



Quad



Quad8

TARGE170

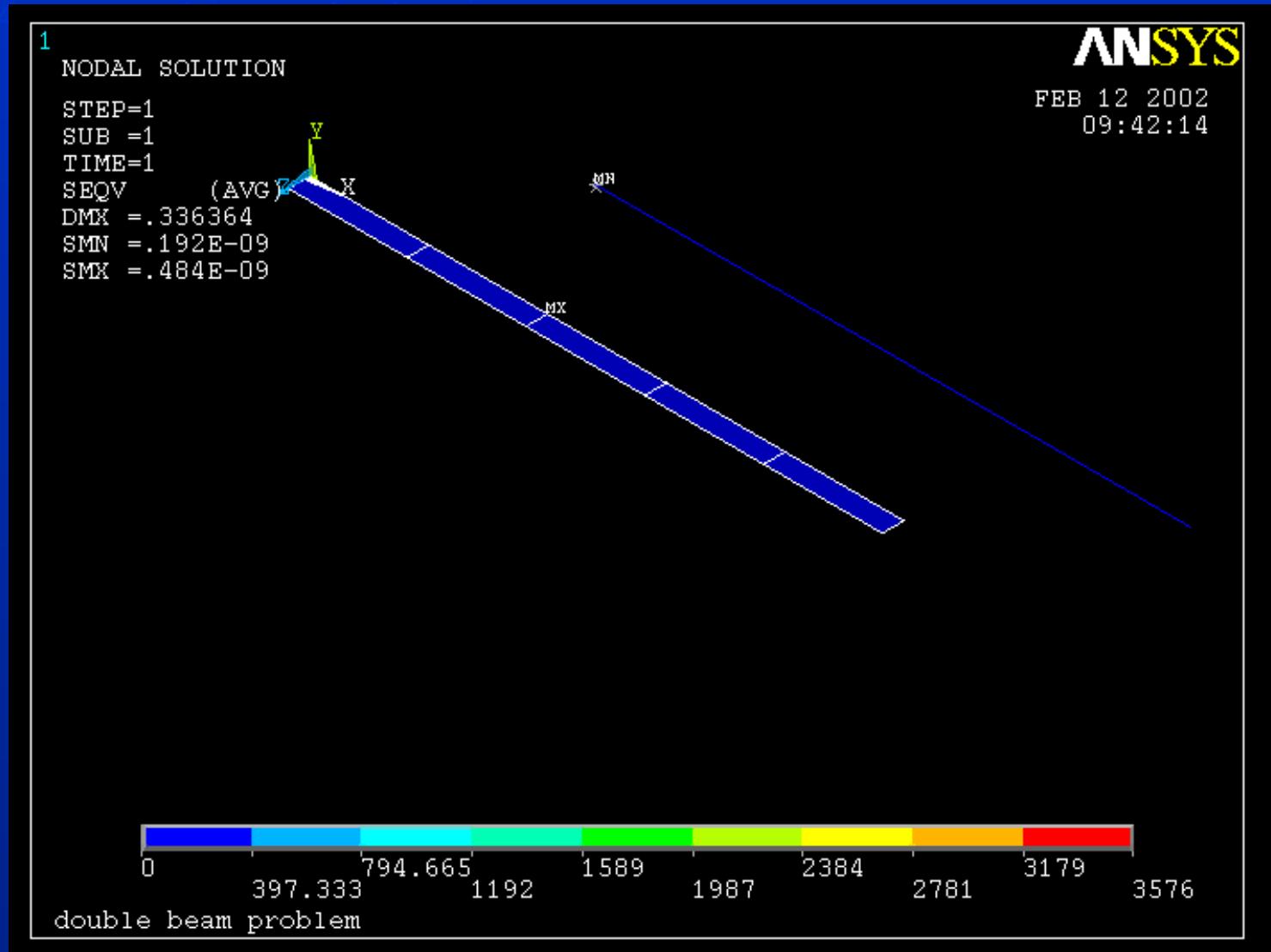
Node-Surf CONT175: Objectives



- Supports dissimilar mesh pattern on both sides of contacting surface that CONTA178 can not handle.
- Solve contact problems when surface-surface contact elements CONTA171-174 have difficulties.
 - Point-surface contact & edge-surface contact
- Replace CONTAC26,48,49.
 - Much less number of elements
 - Better way to treat large frictional sliding
 - 2d/3d rigid-deformable & deformable-deformable contact
 - Midside node for 2d/3d target, 2d contact surface
- This element is typically used only in highly specialized niche applications:
 - In 3D, contact between a line (or sharp edge) and a surface.
 - In 2D, contact between a node (or sharp corner) and a surface.

Double Beams Contact

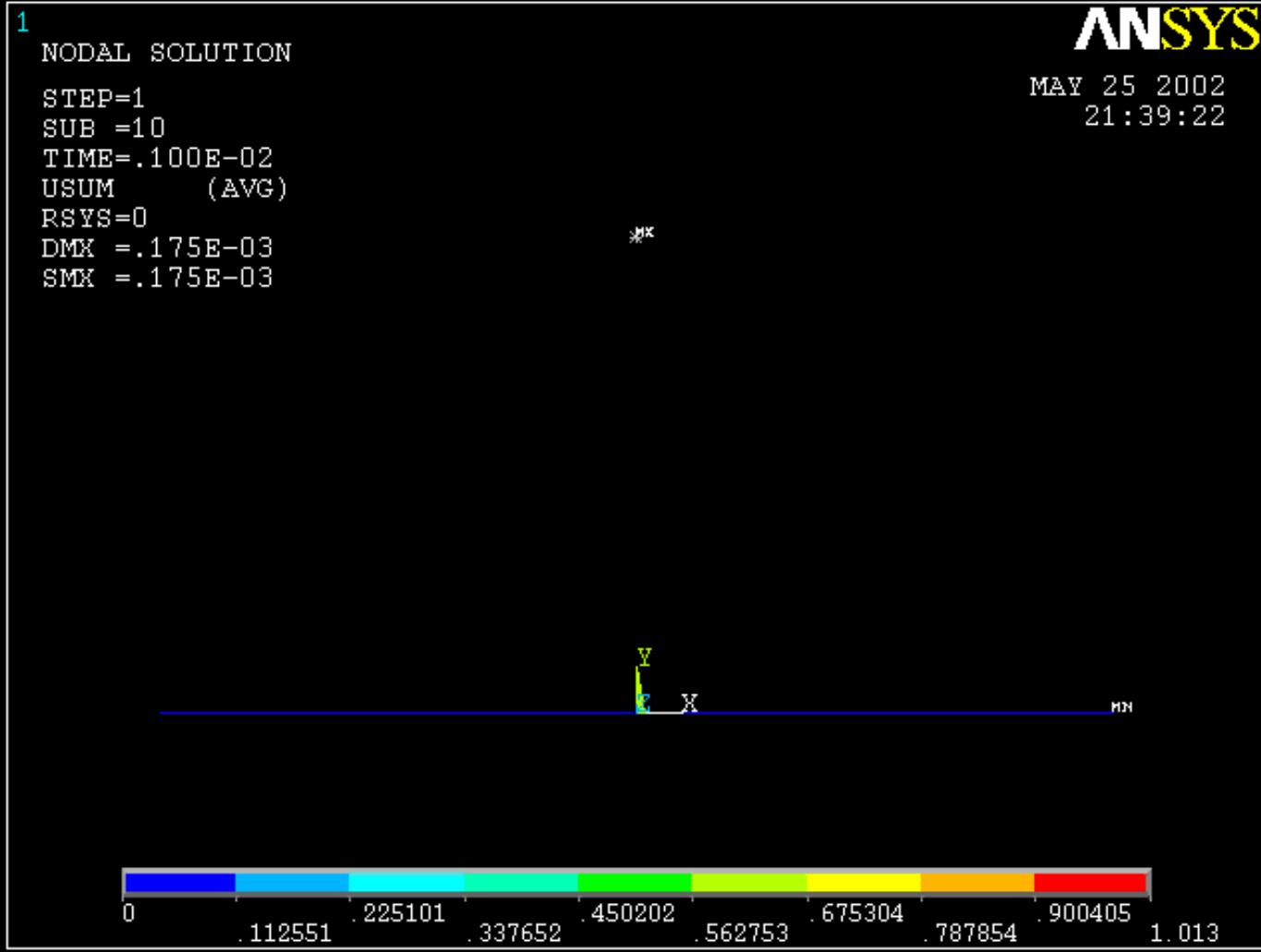
ANSYS



Point Contact

ANSYS

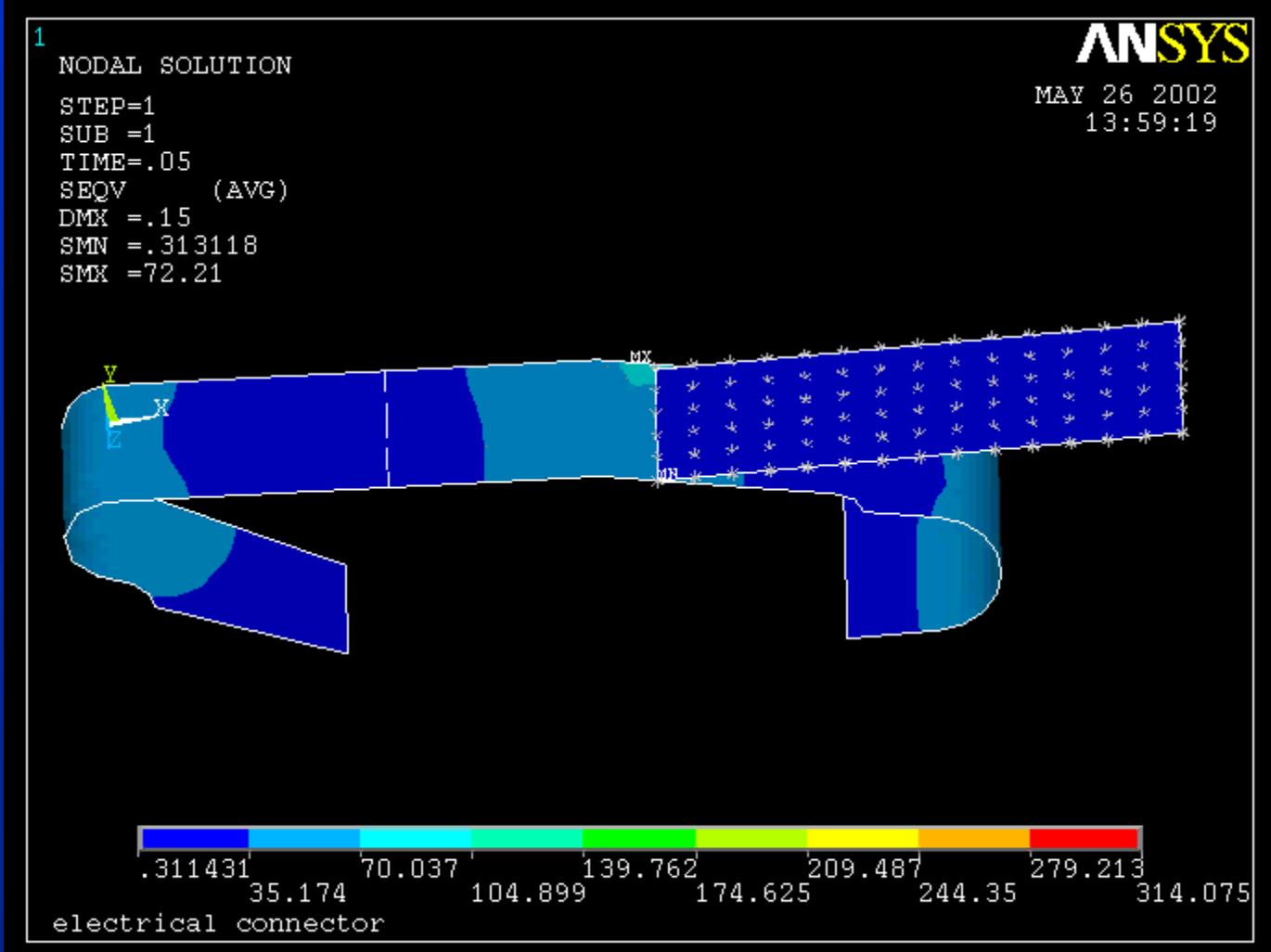
14 sym



Electrical Connector

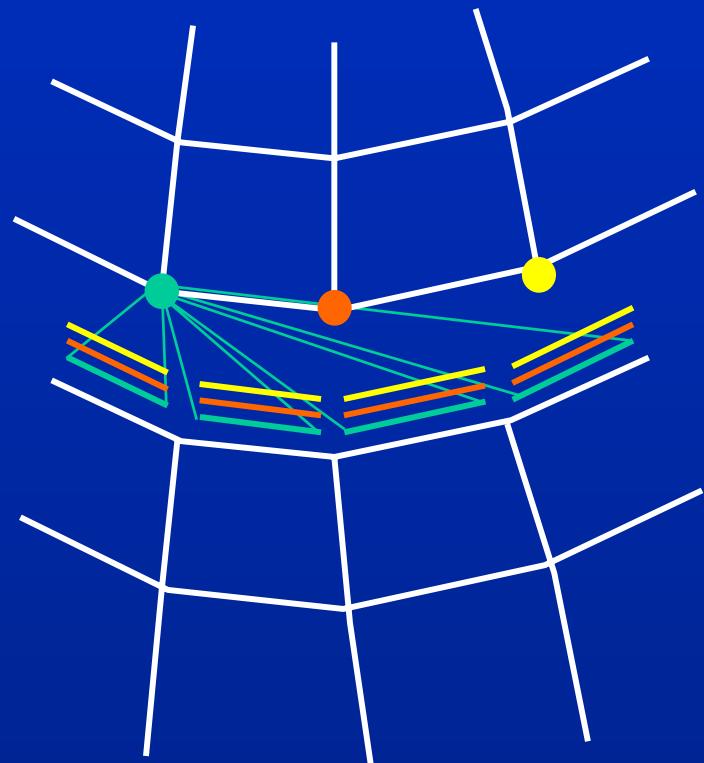
ANSYS

14 sy

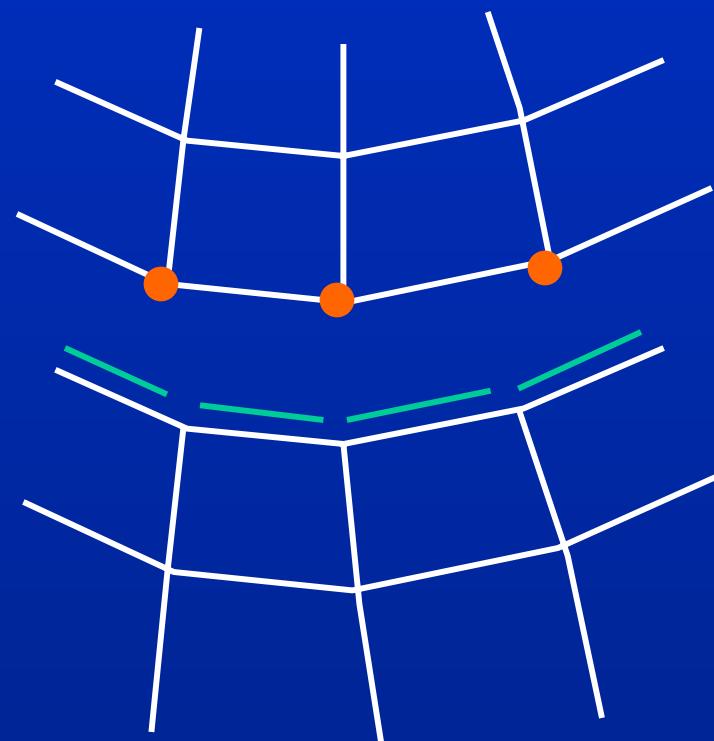


Less Numbers of elements

ANSYS



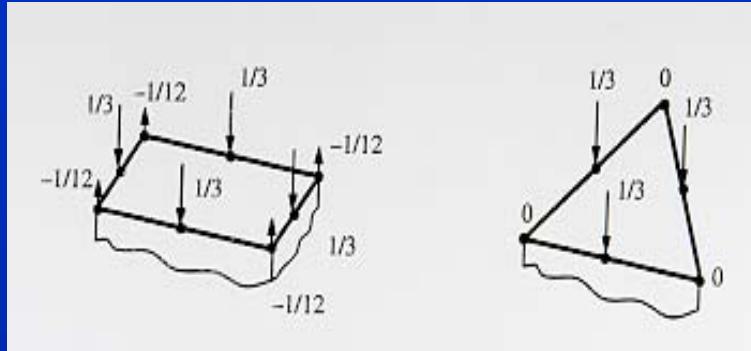
12 CONTAC48
C*T



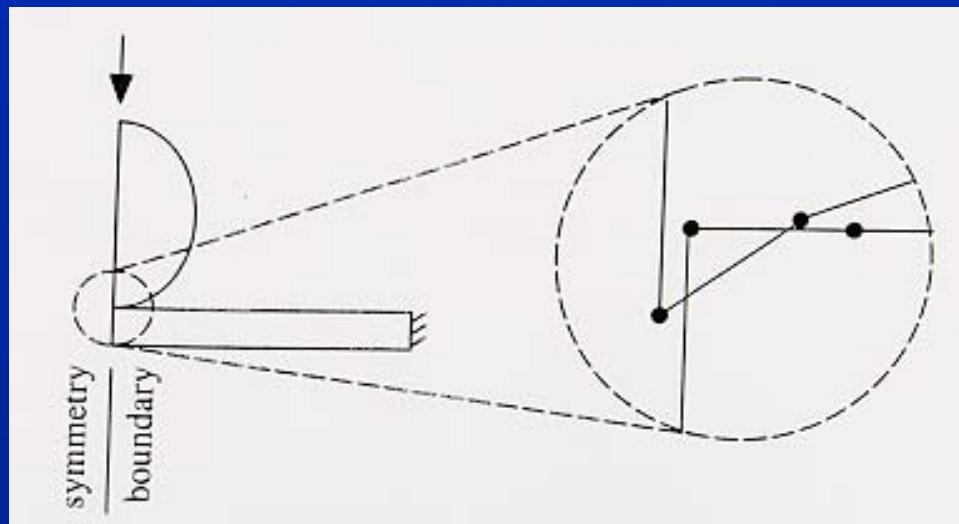
3 CONTA175 + **4** TARGET169
(C+T)

Limitations of Node-Based Contact

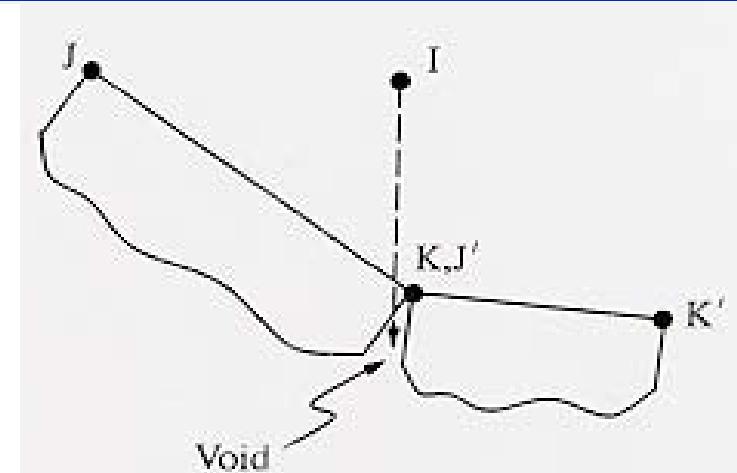
ANSYS



Problem with 3D
higher-order contact
element



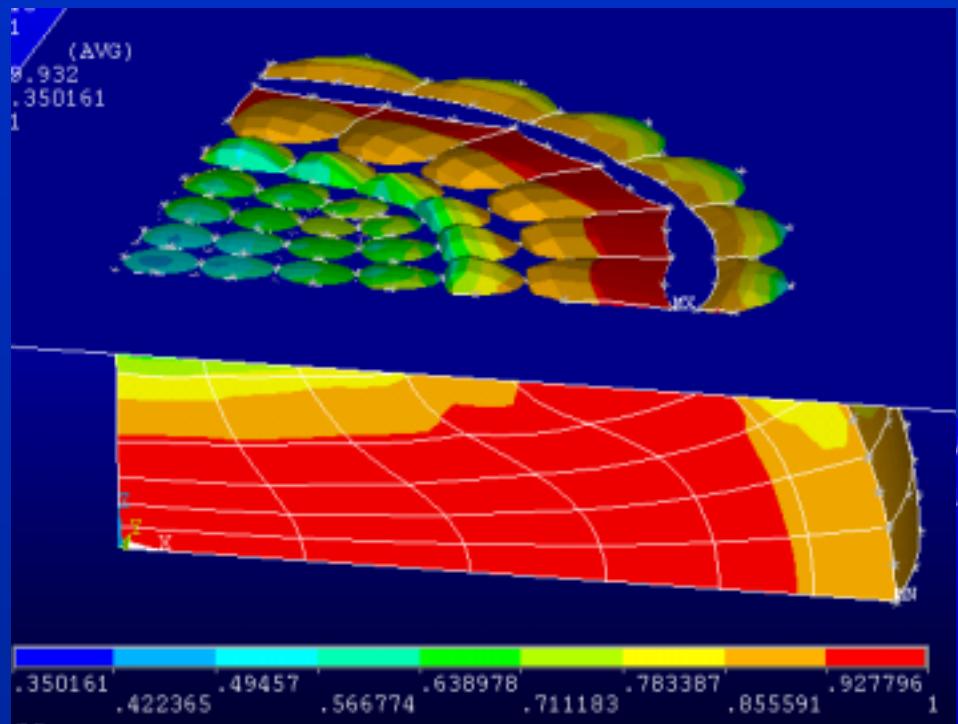
Contact node slips off the
edge of target surface



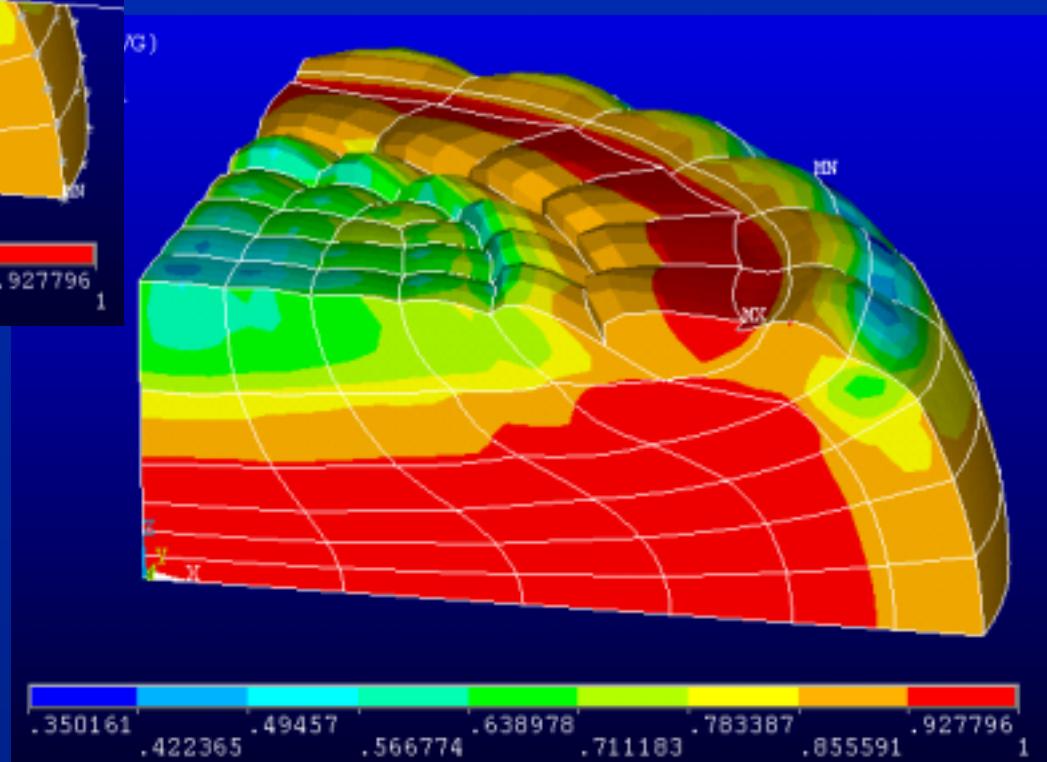
Questionable for
unique penetration

Limitation of Node-Based Contact

ANSYS

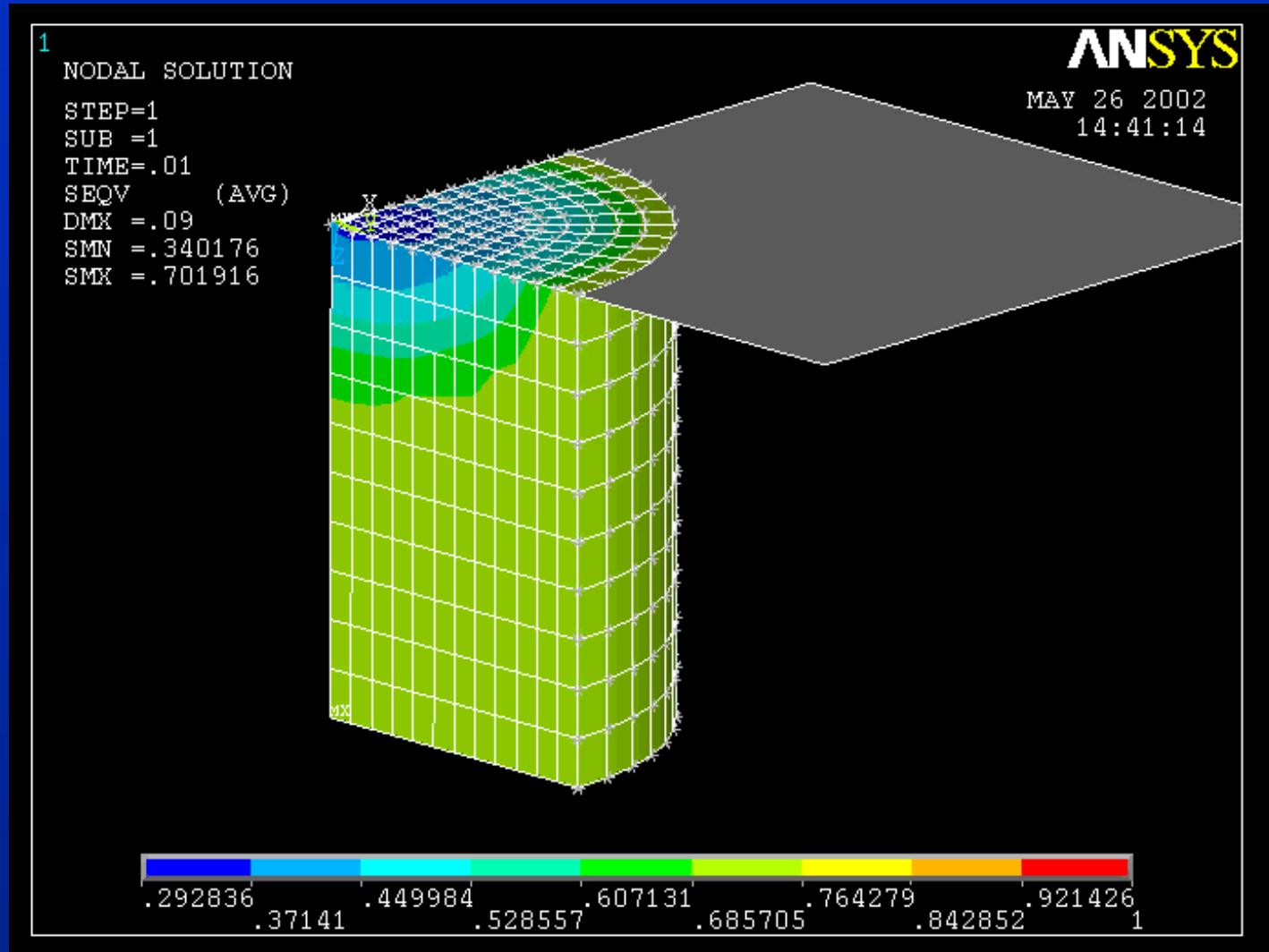


Node-surface handles
20 nodes brick element



3D Upsetting

ANSYS



**Node-surface
handles**
**8 nodes brick
element**

CONTA175: Overview



- The contact nodal forces are assumed to be directly conjugate to the nodal gaps. The contact traction can be recovered by trace the area on contact node.
- Smoothing is done by averaging normal at the target nodes. The normal direction inside of target segment is obtained by interpolation of shape function
- ANSYS automatically extends the target surface to prevent “slip-off”.
- The approach can not support 3D higher order element contact
- The approaches is incapable of passing so-called contact patch tests. It satisfies LBB condition.

CONTA175: Overview

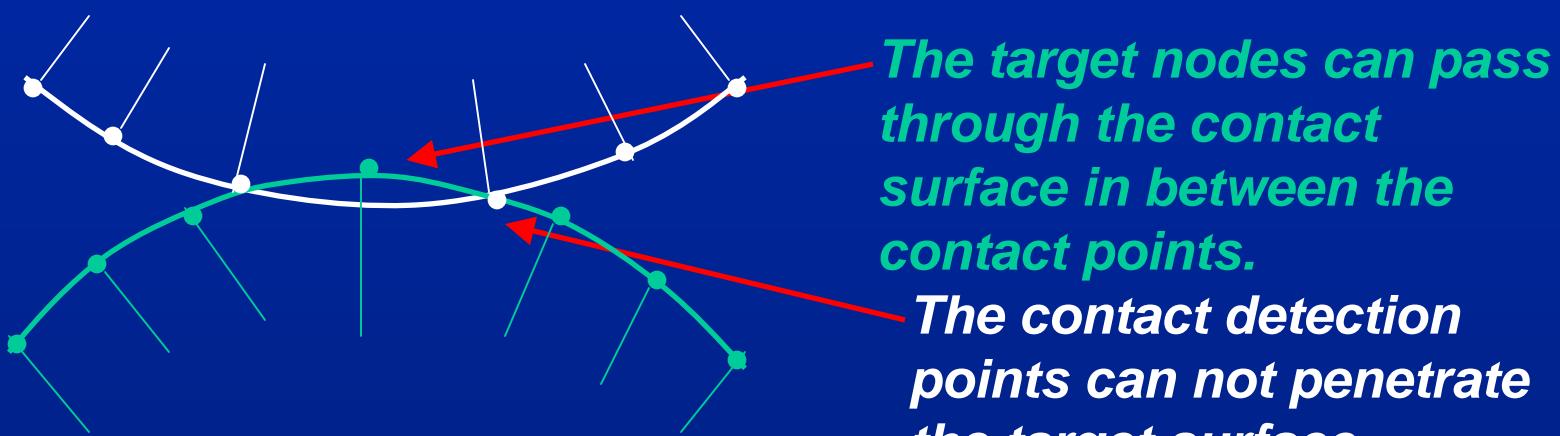


- 2D/3D contact
- Ridge-flexible, flexible-flexible contact
- 2D lower/higher order contact & target
- 3D lower order contact, 3D lower/higher target
- Static, transient, model, linear buckling, harmonic, sub-structuring analysis
- Thermal-electric contact ?

Contact Pair Concept

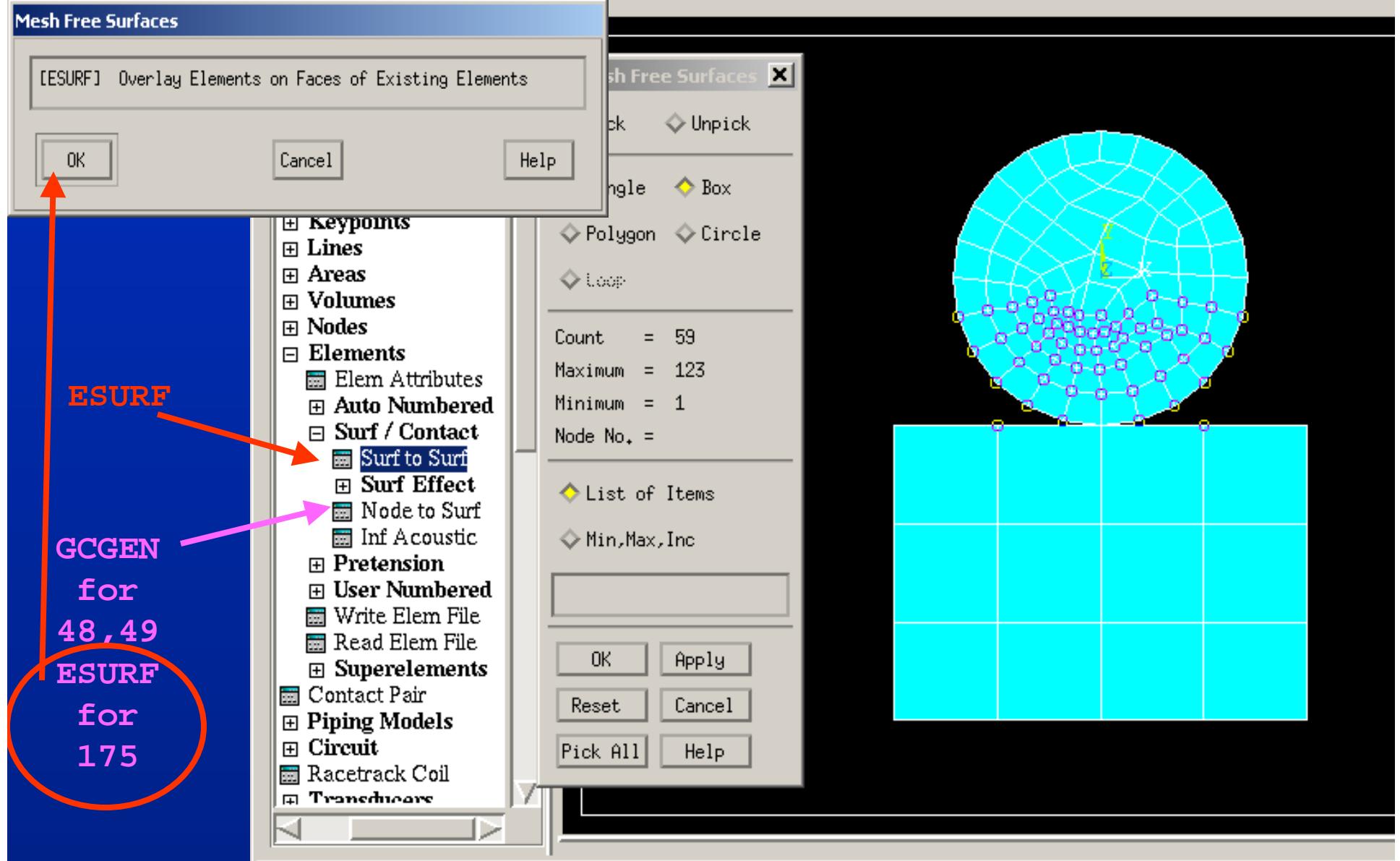
ANSYS

- Target (master) surface - a continuous surface
- Contact (slave) surface
 - A set of discrete contact nodes
 - Contact constraint equation
- Symmetric (two pass) contact
 - Difficulty of contact pressure interpretation
 - Overconstraint of the model



CONTA175 Mesh Tool: ESURF

ANSYS



CONTA175 Mesh Tool: ESURF

ANSYS



Only exterior nodes
are re-selected

CONTA175: Contact Models



- **Contact force based model KEYOP(3) = 0**
 - Gives contact forces as CONTAC48,49
 - FKN, FKT unit: Force/length
 - TCC, ECC depend on element size
 - Pressure → contact force in etable and PLES/PLNS
- **Contact traction based model KEYOP(3)= 1**
 - Gives contact pressure as CONTA171-173
 - FKN, FKT unit: Force/length**3
 - Thermal-electric contact is available (TCC, ECC)
 - Point contact is not supported (auto switching).

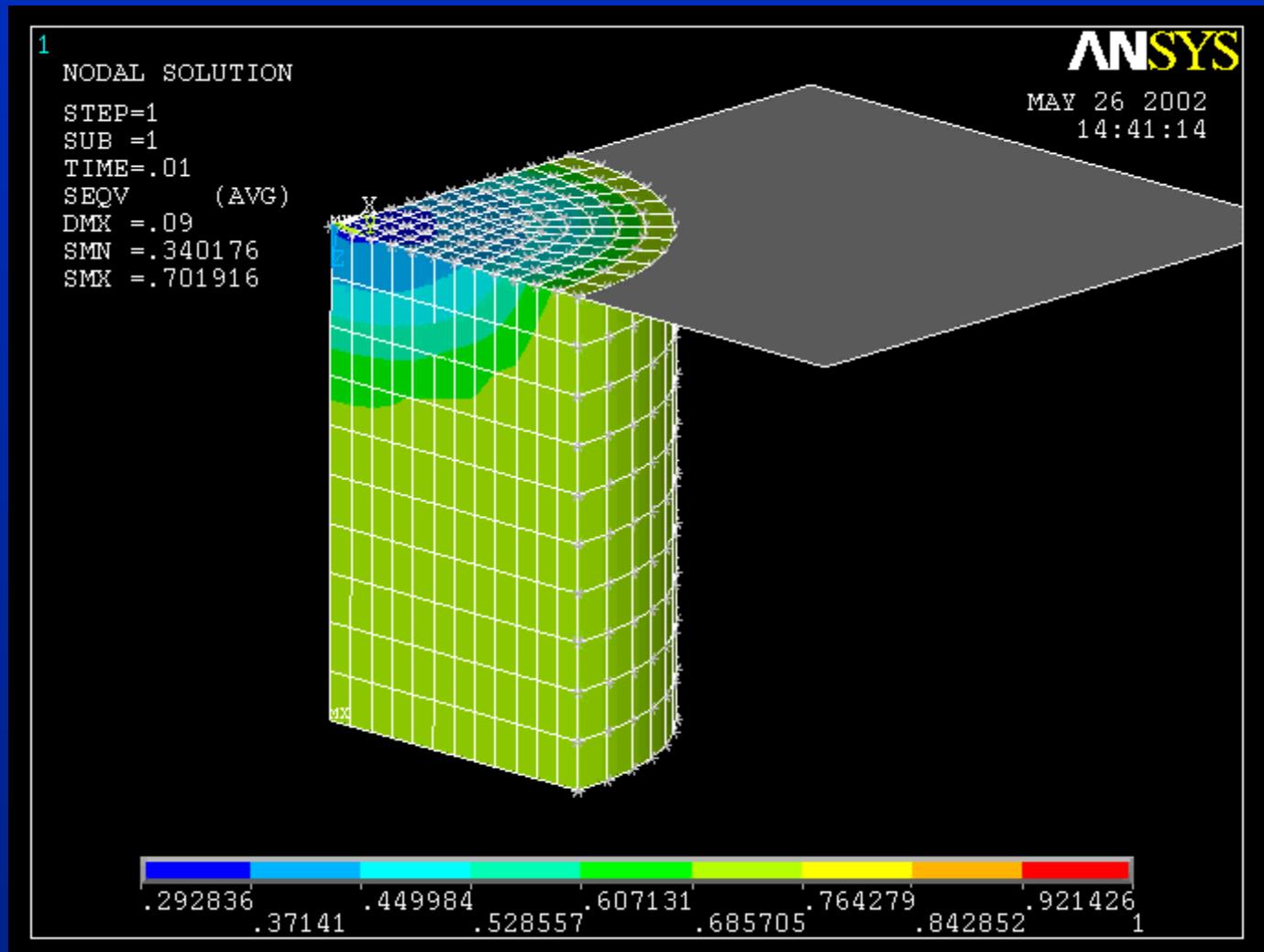
Contact Normal Direction



- Perpendicular to target surface KEYOP(4)=0
 - Target surface smoothing must be done
 - Used if target surface is smoother than contact surface
- Perpendicular to target surface KEYOP(4)=1
 - Contact surface smoothing must be done
- Perpendicular to target surface KEYOP(4)=2
 - Used for shell/beam bottom surface contact
 - Contact surface smoothing must be done
- Smoothing is performed by averaging surface normals connected to the node.

Upsetting

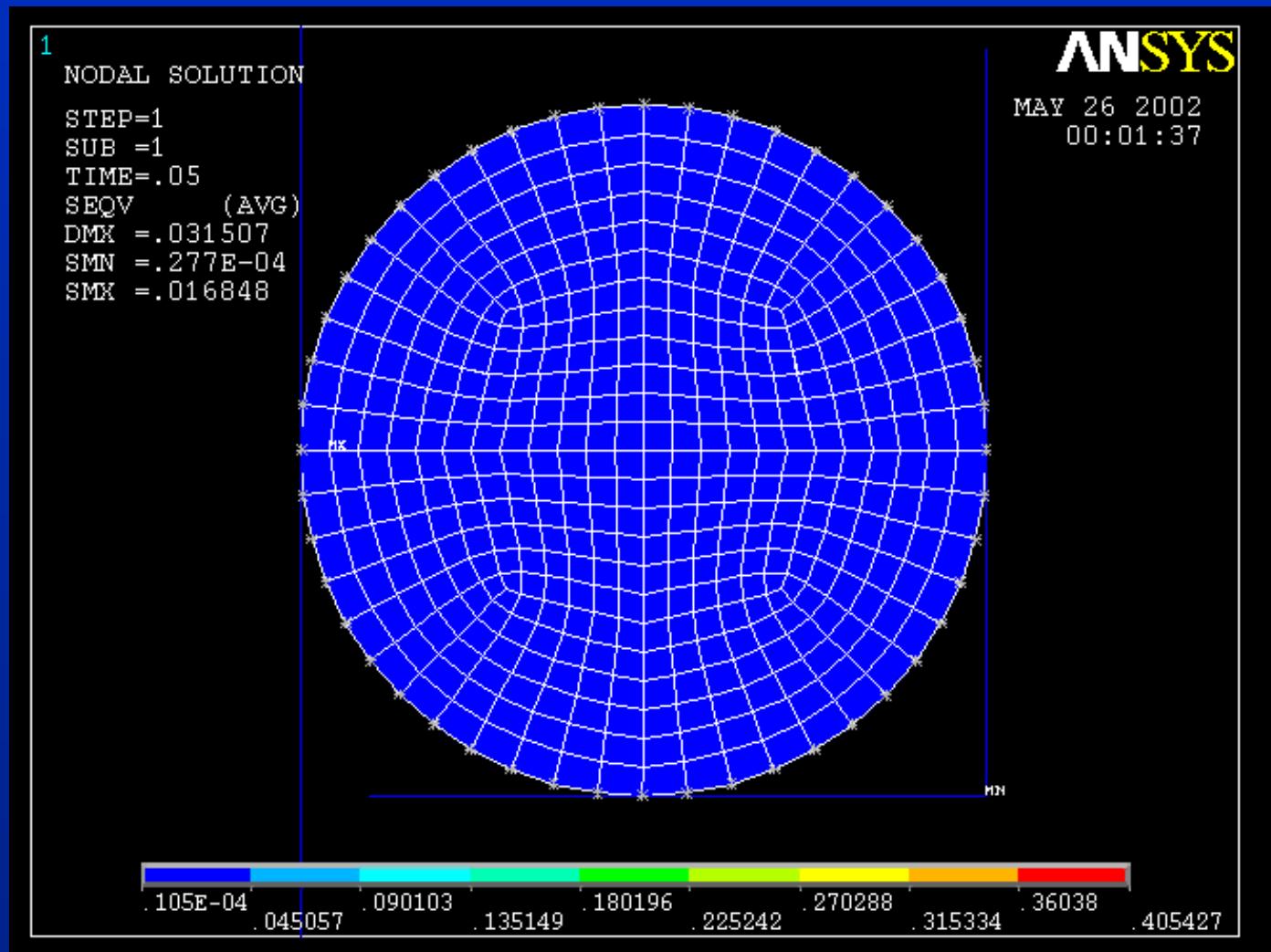
ANSYS



KEYOP(4)=0
Normal to
target surface

O-ring Problem

ANSYS



Double Beams Problem

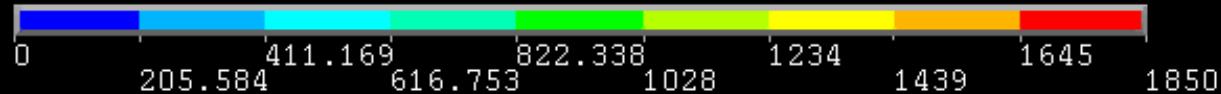
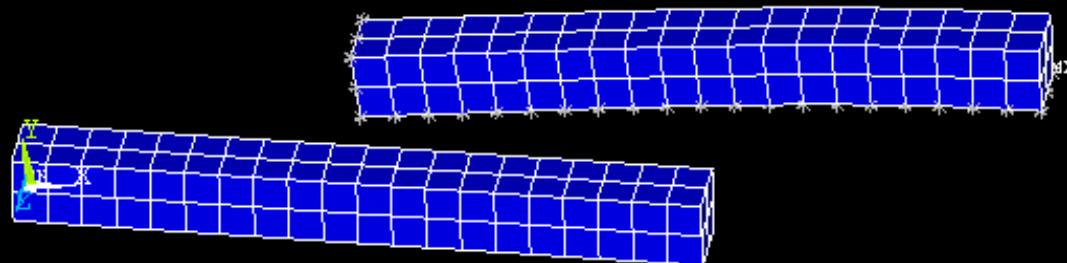
ANSYS

```
1  
NODAL SOLUTION  
STEP=1  
SUB =1  
TIME=.1  
SEQV (AVG)  
DMX =6.855  
SMX =140.513
```

ANSYS

MAY 26 2002
16:09:30

Different normal
Definition gives
Different answers



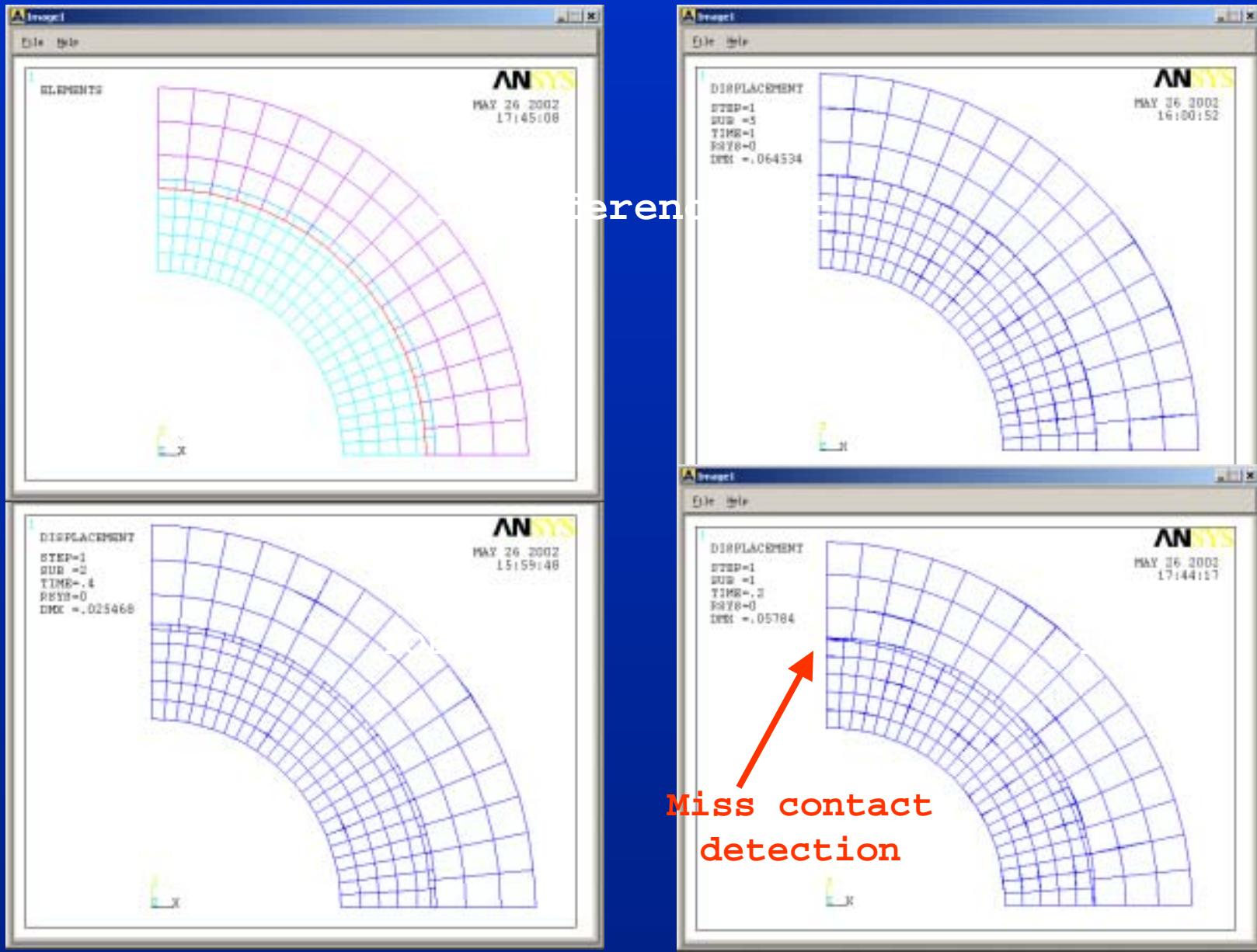
CONTA175: Target Edge Extension



- **Auto extension of target surface (TOLS)**
 - Prevent from slipping of the target edge
 - Defaults 2 (2%) for nlgeom,on; 10 for nlgeom,off; 20 for interference fit

Target Edge extension factor

ANSYS



CONTA175: Postprocess



- Contact quantities can not be clearly visualized (as opposed to CONTA171-173).
- Etable items and Sequence Numbers are similar to CONTA171-173
 - Easily switch element types
 - Waste spare space
- Items in PLES, PLNS for contact force based model
 - PRES: contact nodal force
 - SFRI: contact frictional force

Name	Item	E	I	J
PRES	SMISC	5	1	2
SFRIC	SMISC	-	3	4
STAT <u>1</u>	NMISC	19	1	2
OLDST	NMISC	-	3	4
PENE <u>2</u>	NMISC	-	5	6

Node-Surf CONTA175: Limitation



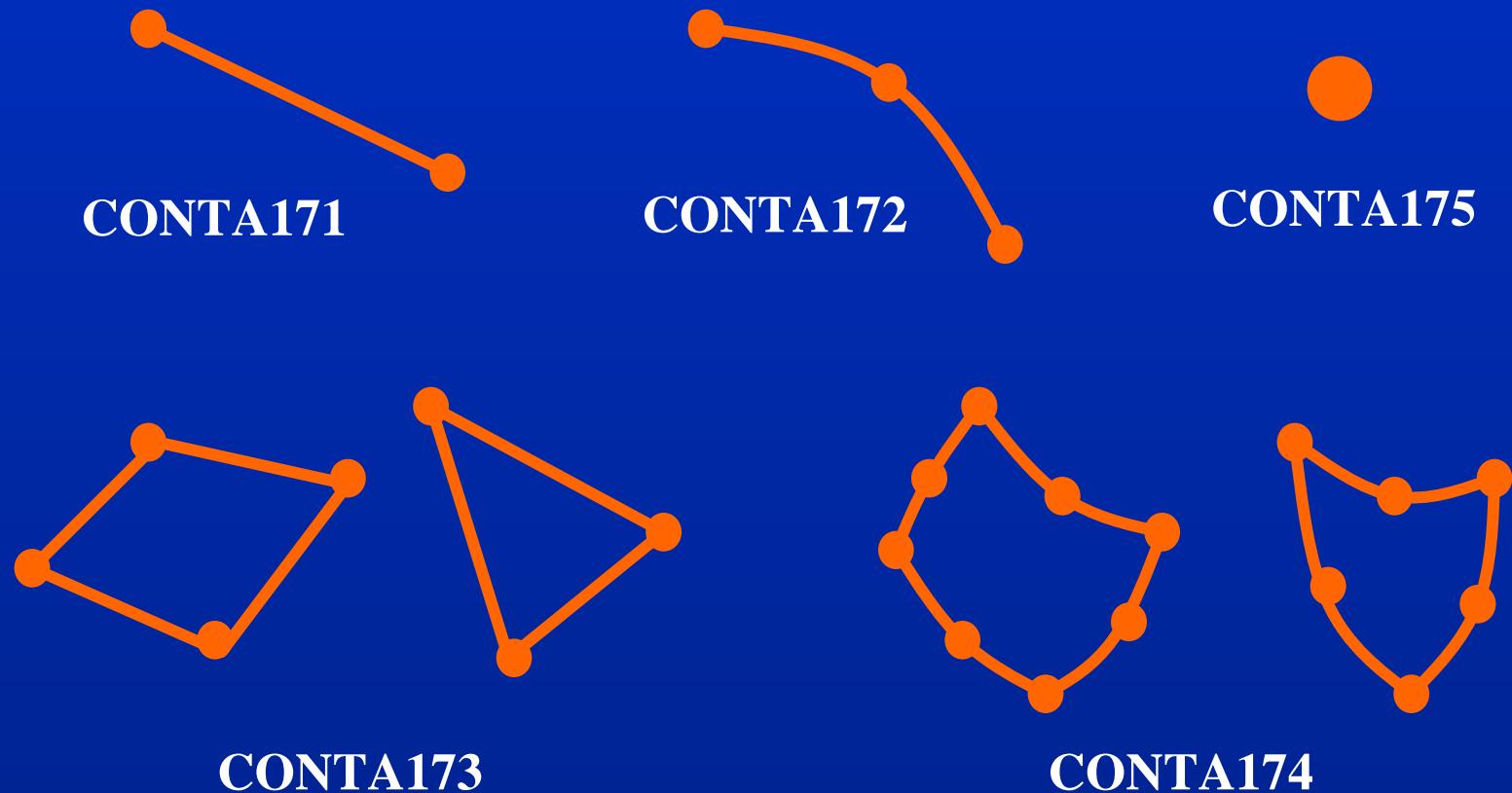
- Does not support 3D higher order element contact
- Requires 8 nodes hexahedron
- Contact results can not be visualized
- The approaches is incapable of passing so-called contact patch tests.

Surface-Surface Contact Element CONTA171-174

A New Revolution Technology

Contact Elements

ANSYS



Target Segments

ANSYS



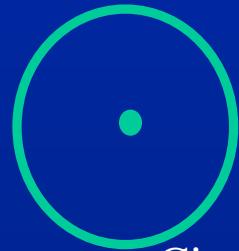
Line



Parabola



Arc

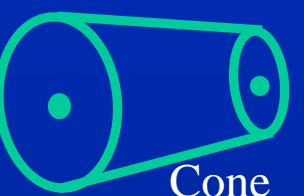


Circle

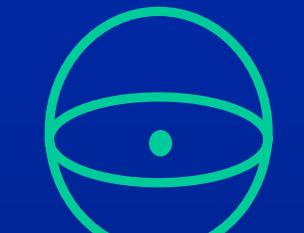
TARGE169



Cylinder

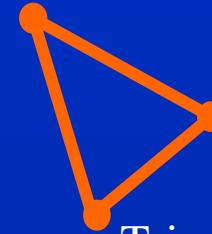


Cone

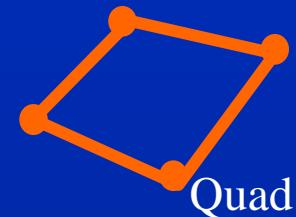


Sphere

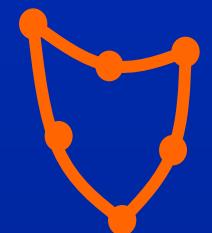
Pilot node



Tri



Quad



Tri6



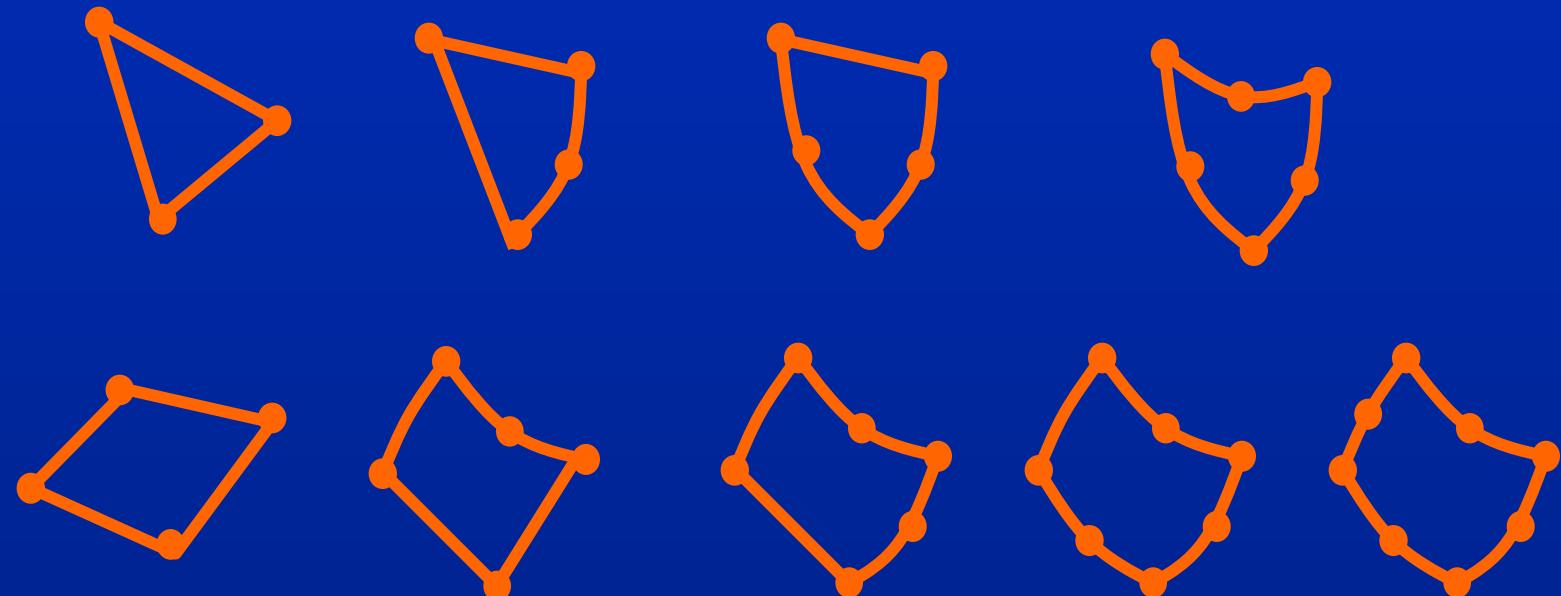
Quad8

TARGE170

Contact/Target Degenerated Shape

ANSYS

- Fully support 3D 2nd order of contact and target elements with any kind of dropping mid-side nodes.



Surface-to-surface elements



- The surface-to-surface contact elements are the most versatile contact elements in the ANSYS program.
- Because they are robust, feature-rich, and user-friendly, they have become the contact element of choice for most ANSYS users.

Surface-to-surface elements: Overview



- The surface-to-surface elements are the most widely used contact elements in ANSYS, due to the many advantages that they have over the other contact elements:
 - Compatible with both lower order and higher order elements.
 - Support large deformations with significant amounts of sliding and friction efficiently.
 - Provide better contact results (easier to postprocess contact pressure and frictional stresses).
 - Can account for shell and beam thickness, as well as shell thickness changes.
 - Semi-automatic contact stiffness calculation.
 - “Pilot node” control of rigid surface.
 - Intelligent default settings, Contact Wizard (easy to use).
 - Multiphysics contact capability.
 - Numerous advanced options for overcoming difficult problems.

Surf-Surf Contact - Objectives

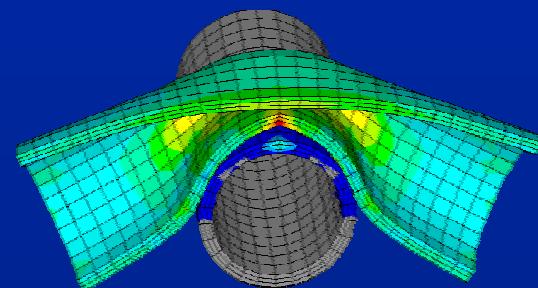
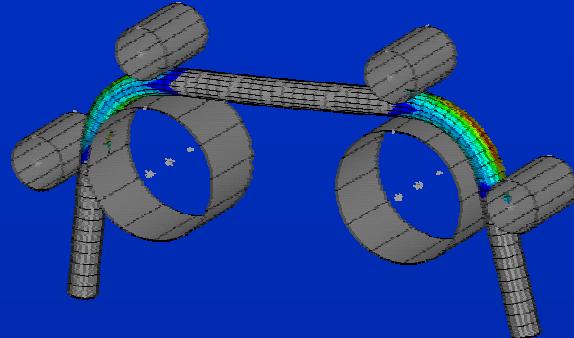


- Extend family of ANSYS contact elements
 - Surf-surf instead of node-node, node-surf contact
- Provide higher-order contact/target elements
 - Mesh generation of 10 nodes tetrahedral is not an issue
 - Represent curve surfaces (non-faceted)
- Provide general thermal/electric contact analysis capability
- Support unmatched mesh pattern on contacting bodies
- Easy of use and intelligent default settings
- Solve real world problems

ANSYS Contact - Overview

ANSYS

- **Rigid-flex**
 - Bodies of vastly different stiffness
 - Steel against rubber seals
- **Flex-flex**
 - Bodies of comparable stiffness
 - Metal contacting metal
- **Self contact**
 - Body folds over itself
 - Column buckling
- **Large sliding with friction for all**



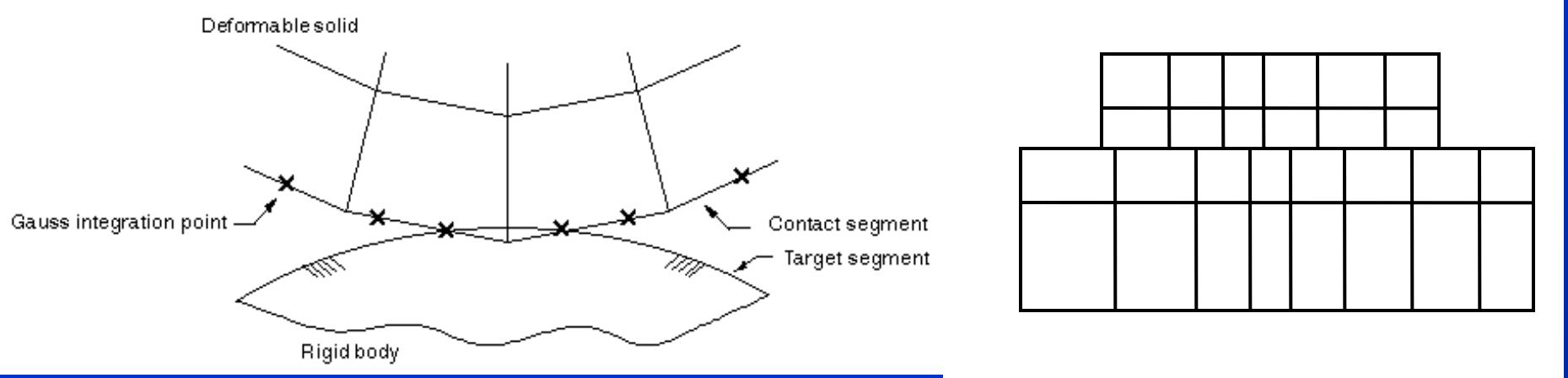
Surface-Surface Contact: Overview



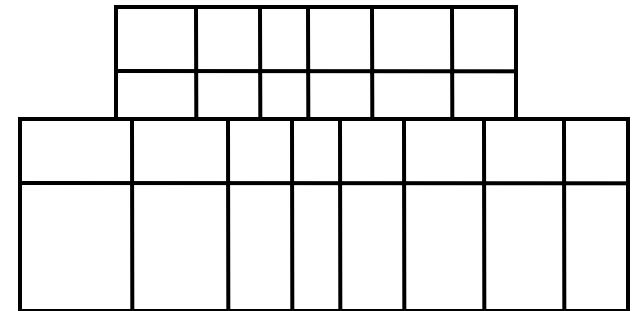
- The formulation is given in terms of contact tractions
- Transmission of pressure across contact element surface is basic to contact problem
- Pressure is applied to element faces by using element shape functions to calculate the equivalent consistent nodal forces
- The use of surface traction approach is recommended as the influence of element size is eliminated which supports well for thermal/electric conductance analysis.
- Patch tests cannot be expected to be passed for arbitrary discretization and for arbitrary element orders.

Surf-Surf Contact - Overview

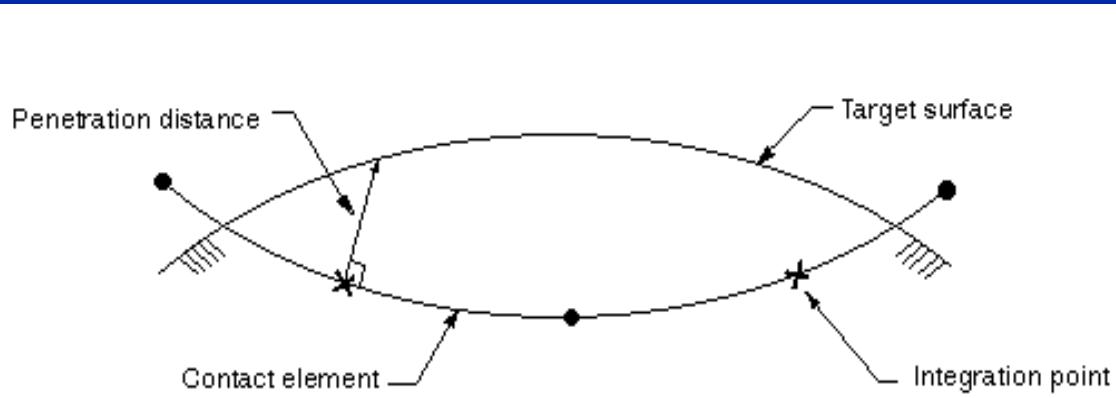
ANSYS



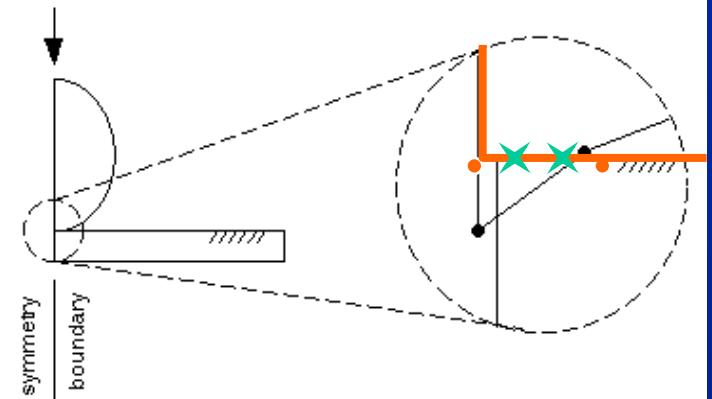
Consistent nodal forces for higher-order elements



Consistent thermal interface definition



Unique penetration, No smoothing of target surface

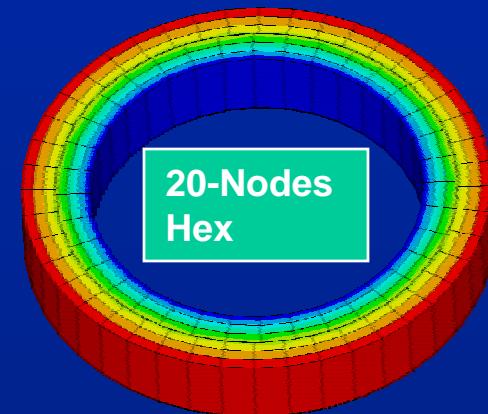
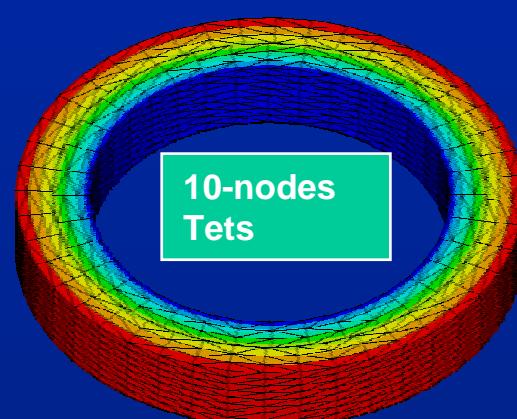
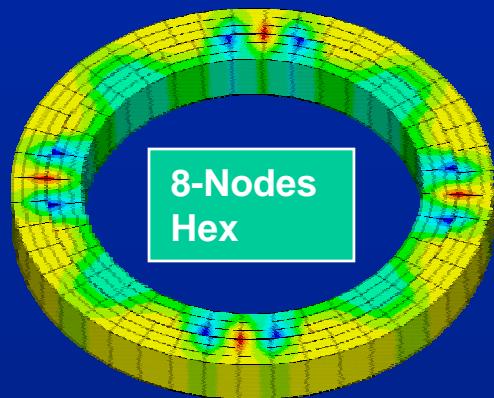


No contact node “slips” off the edge

Surf-Surf Contact - Overview

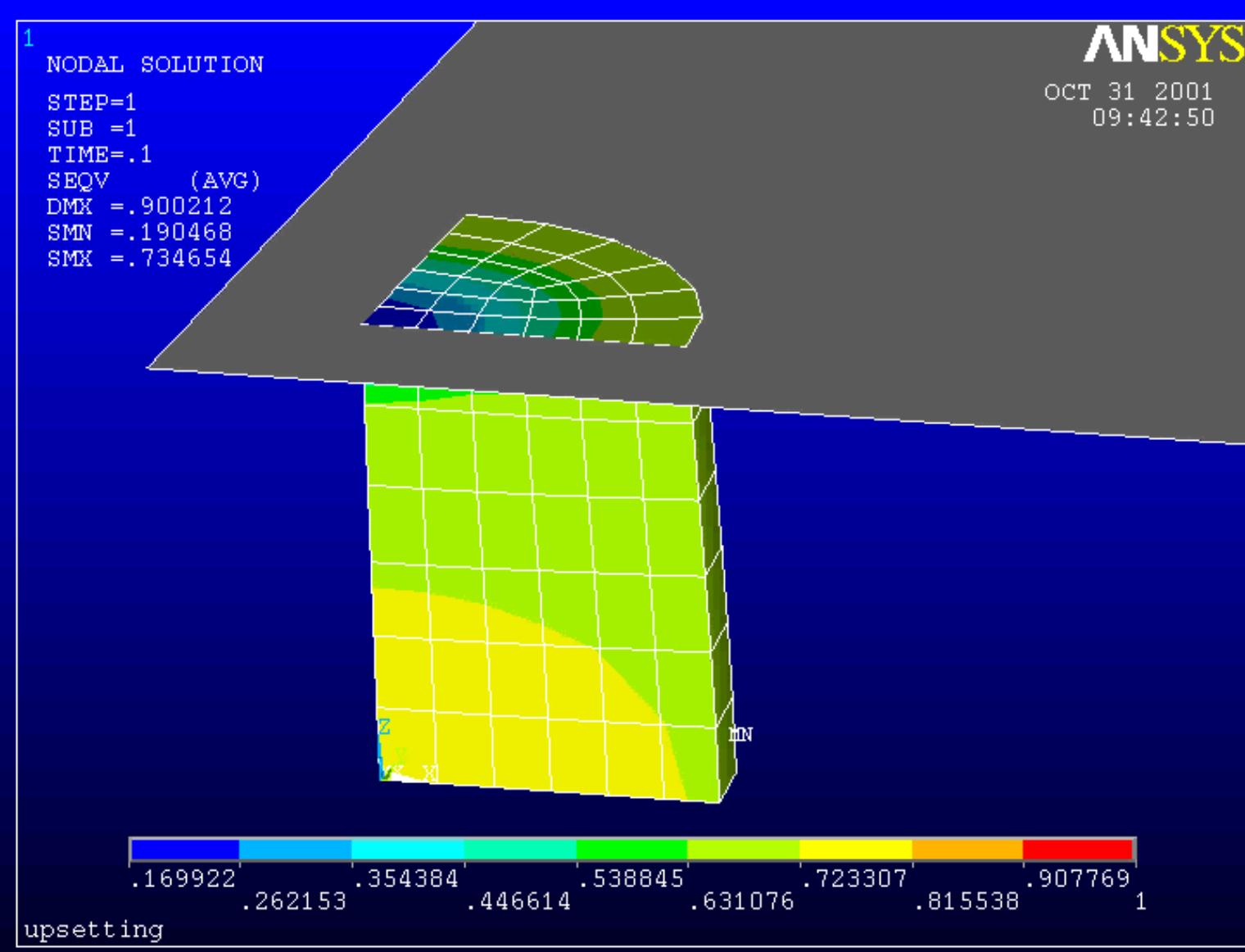
ANSYS

- **Quadratic order contact/target elements**
 - Comparatively stiff behavior associated with 4 nodes tet
 - Fully Automated mesh generation
 - 10 nodes tetrahedron is not an issue
 - 8 nodes Hexahedron is challenging
 - Represent curved surfaces (**non-faceted**)



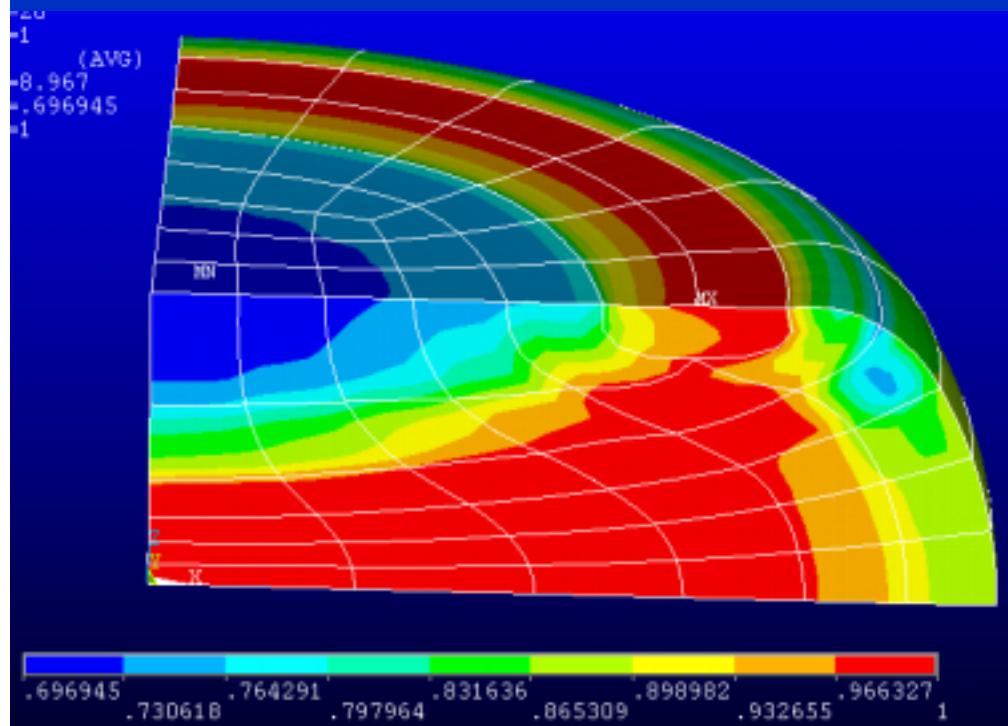
Surf-Surf Contact - Overview

ANSYS



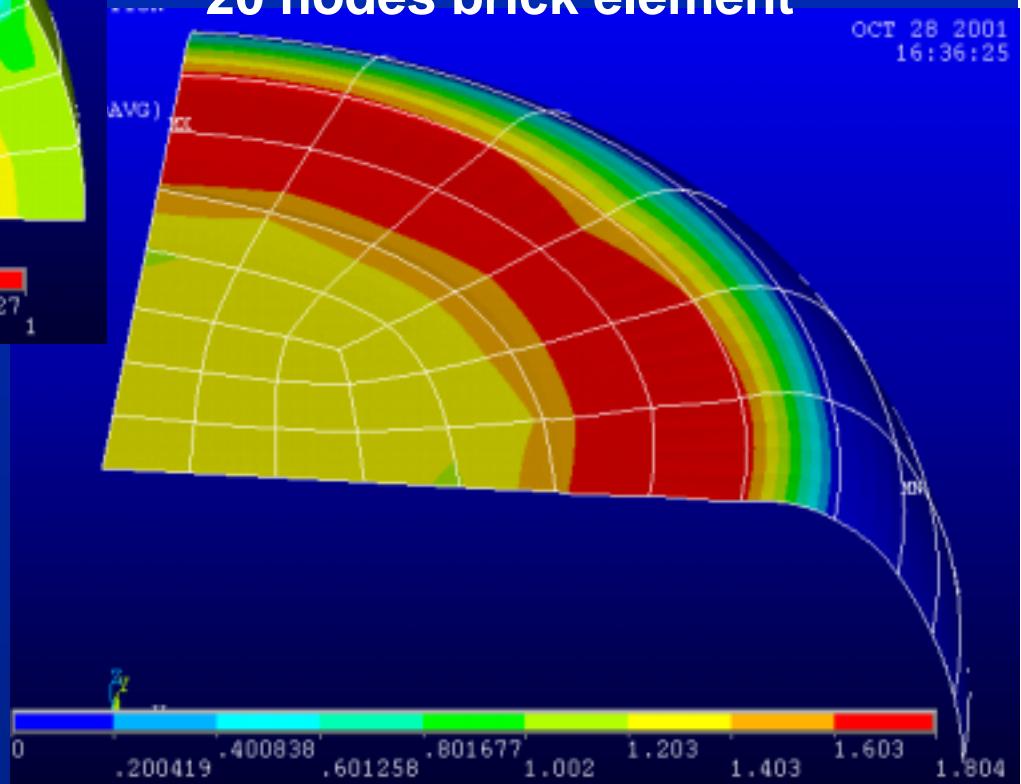
Surf-Surf Contact - Overview

ANSYS



Surface-surface handles

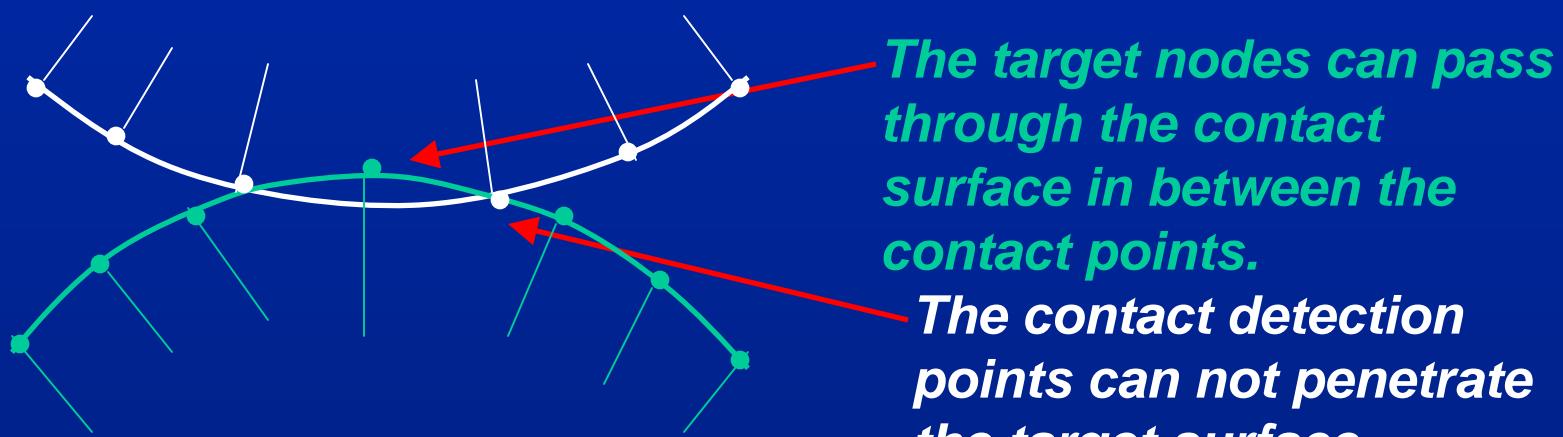
20 nodes brick element



Contact Pair Concept

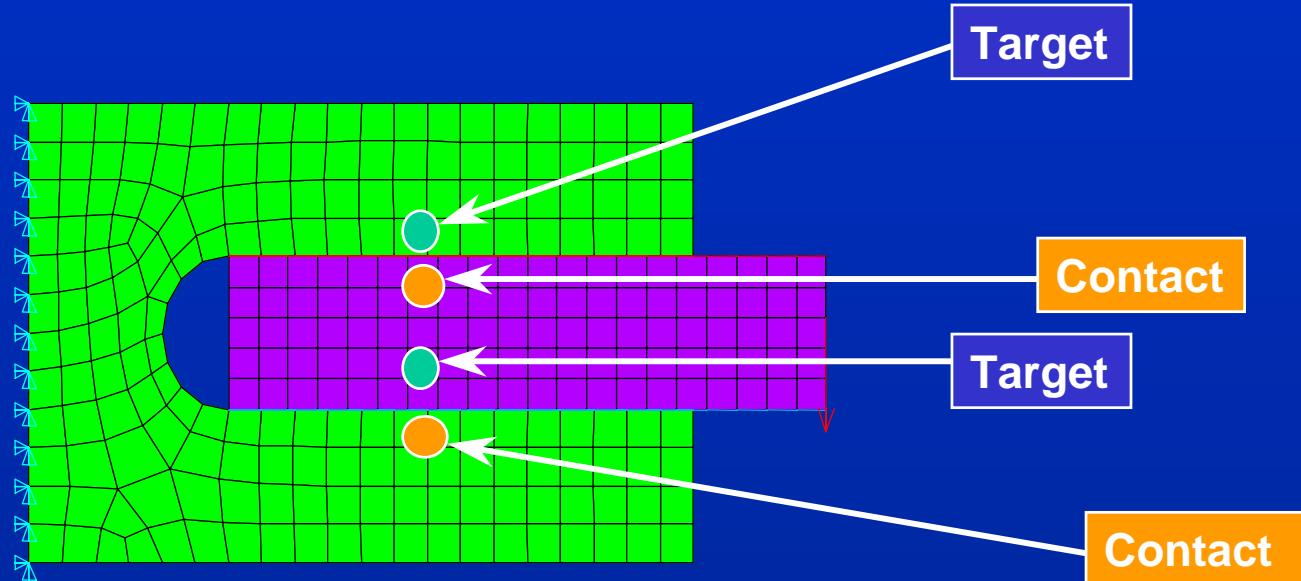
ANSYS

- Target (master) surface - a continuous surface
- Contact (slave) surface
 - A set of discrete contact points (element quadrature points)
 - Contact constraint equation
- Symmetric (two pass) contact
 - Difficulty of contact pressure interpretation
 - Overconstraint of the model



Contact Pair Concept

ANSYS



Contact surface:

- Convex surface
- Fine mesh surface
- Softer surface
- Higher order surface
- Smaller surface

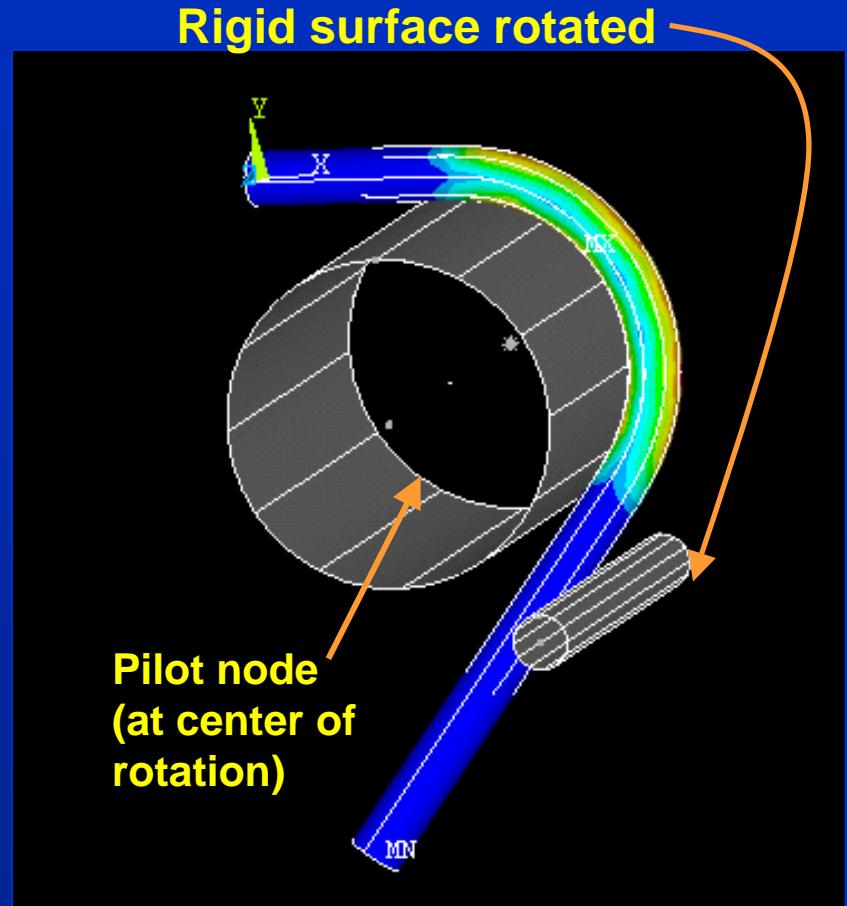
Target surface:

- Concave/flat surface
- Coarse mesh surface
- Stiffer surface
- Lower order surface
- Larger surface

Pilot Node

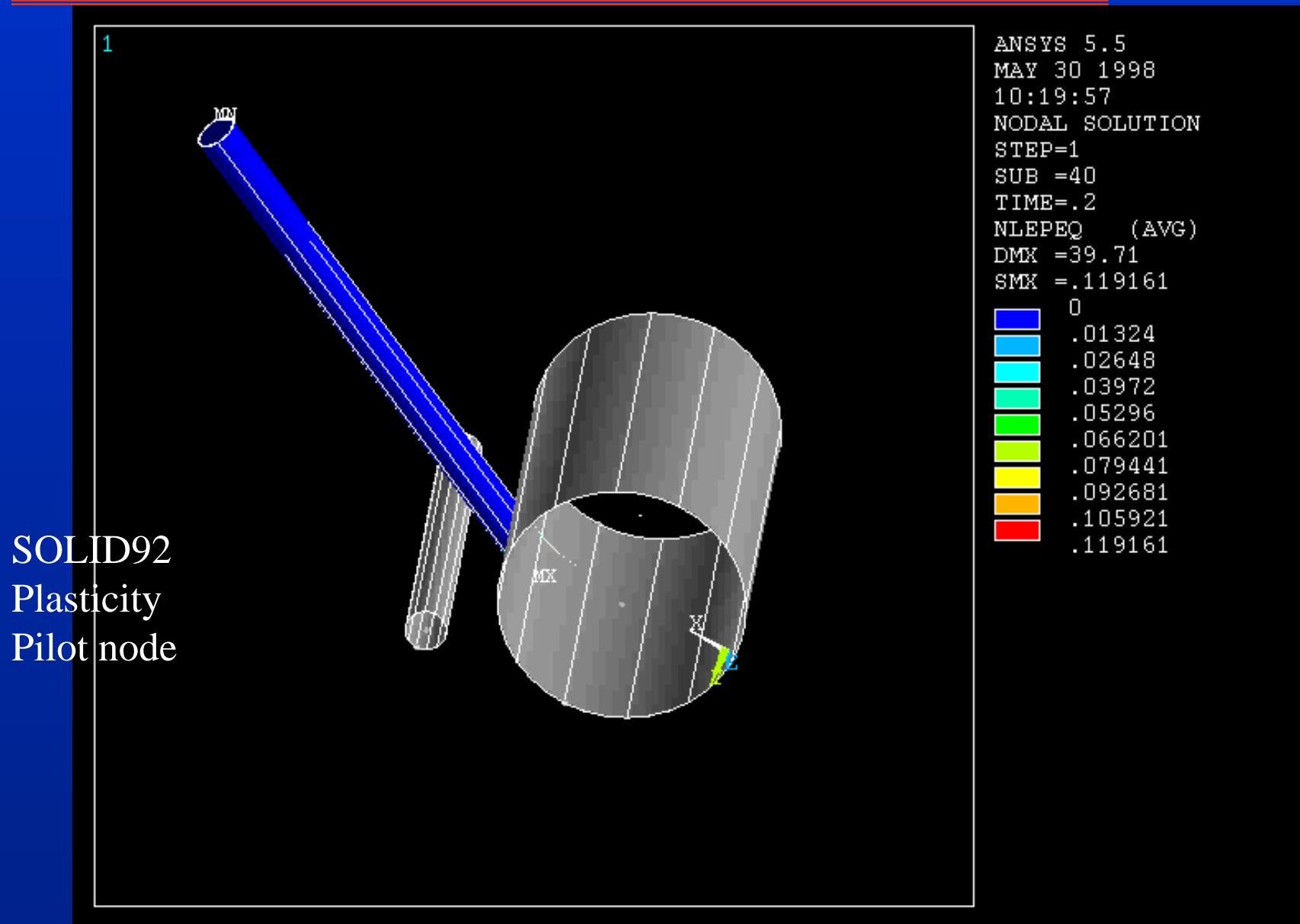
ANSYS

- Governs the motion of target surface
 - Force/displacement
 - Rotation/moment
 - Temperature/voltage
- Characterizes heat flow
 - -: net heat loss
 - +: net heat gain
- Can be at any location
- Connects to other element
 - Mass element



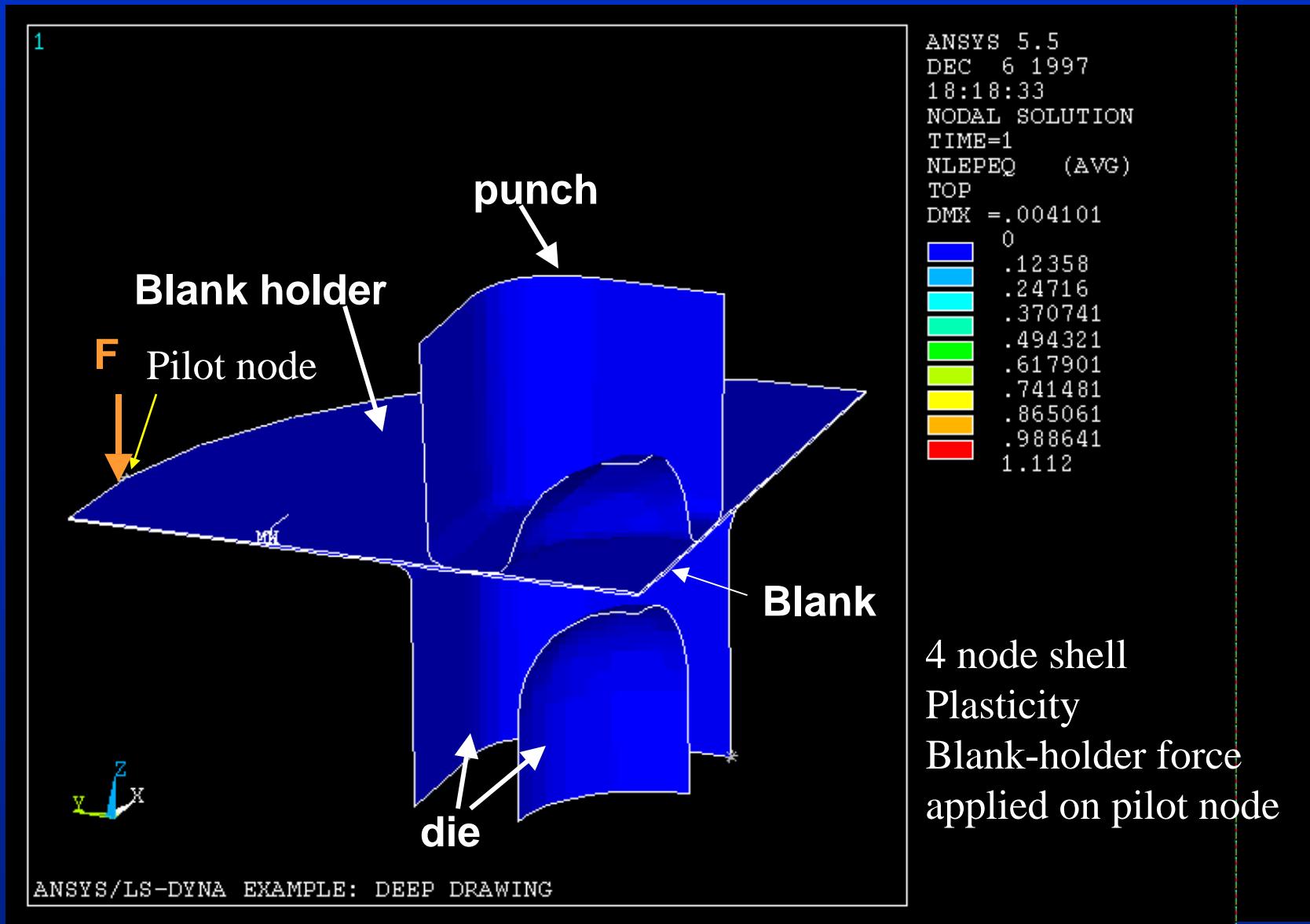
Application: Wire Bending

ANSYS



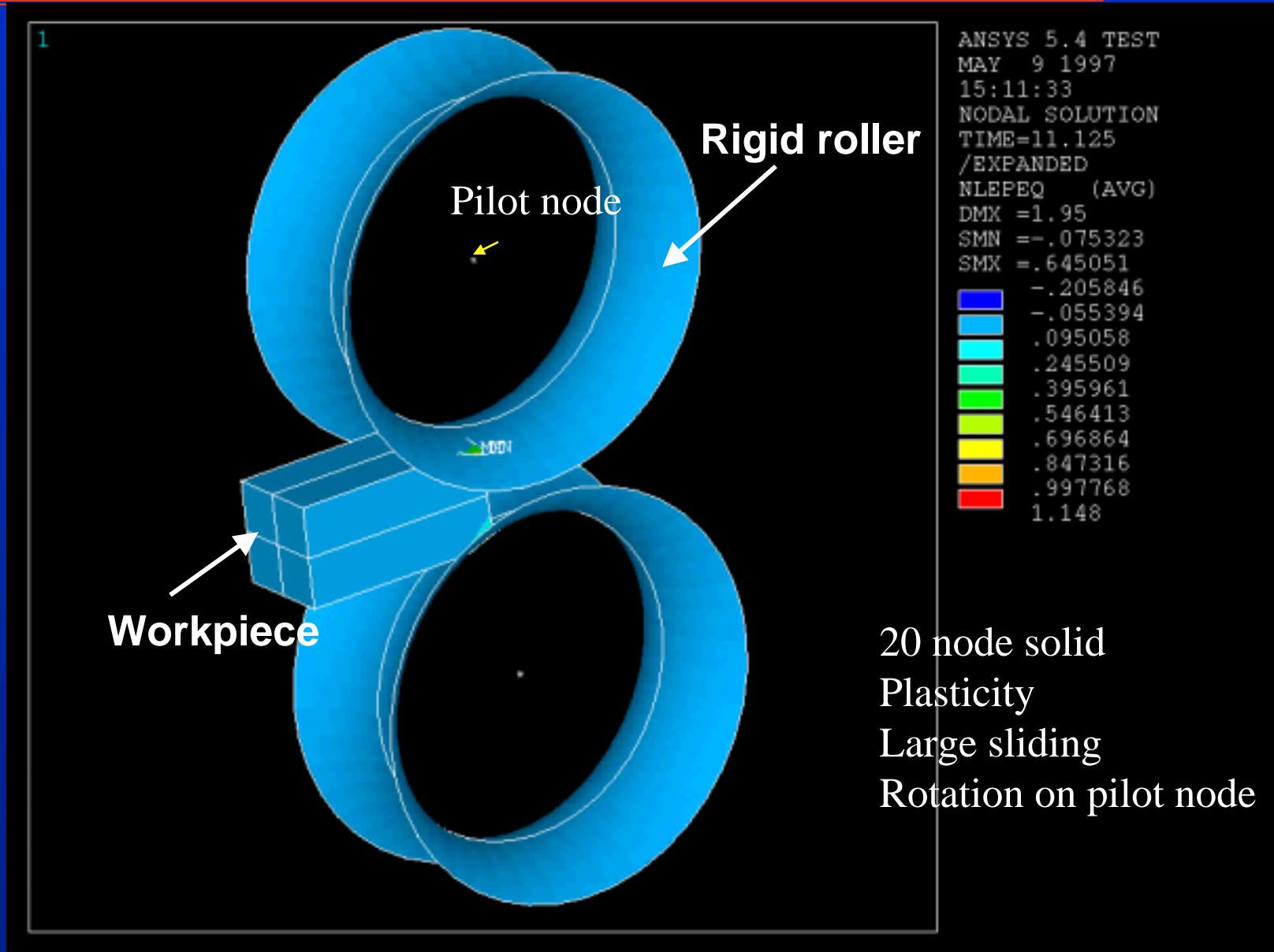
Application: Deep Drawing

ANSYS



Application: Shape Rolling

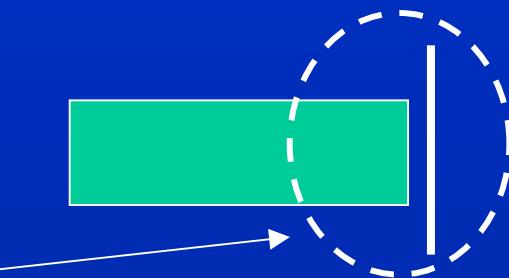
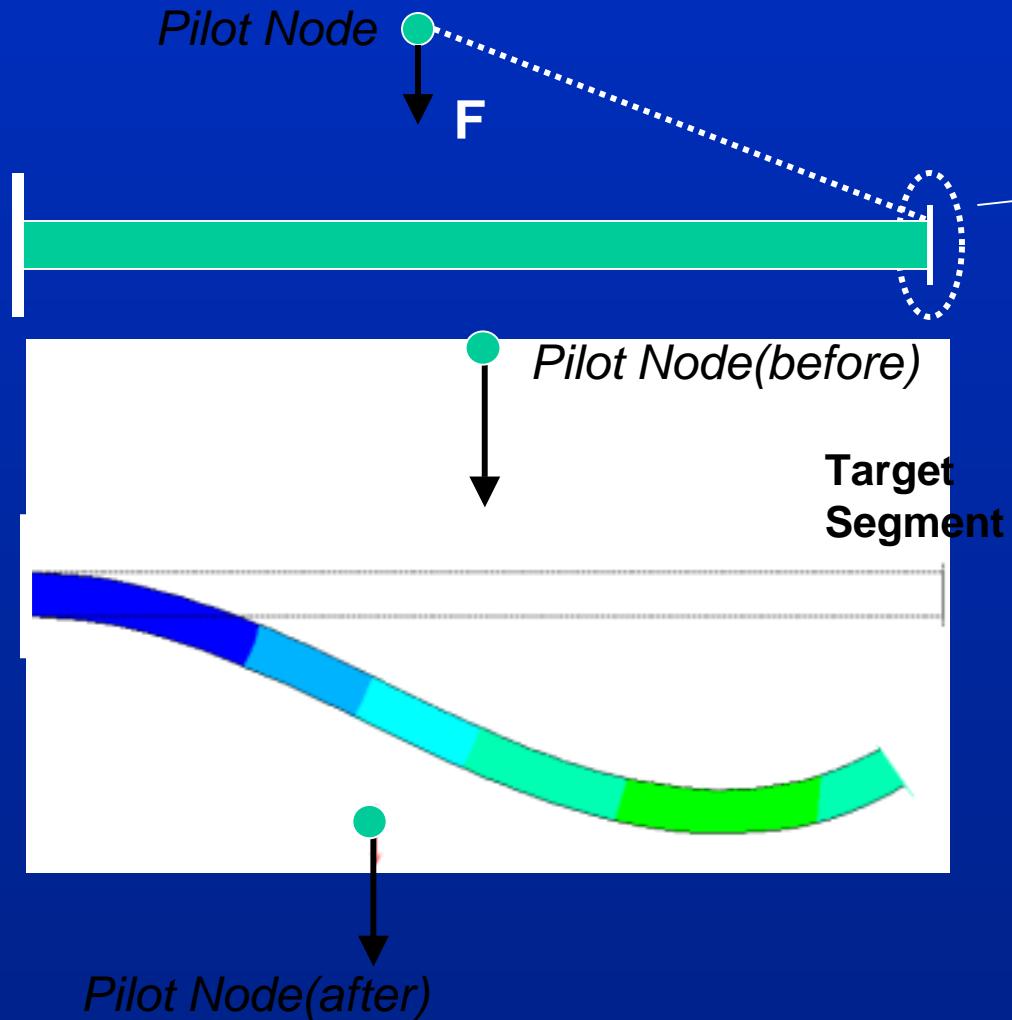
ANSYS



Pilot Node

ANSYS

Pilot node:KEYOPT(2)=1



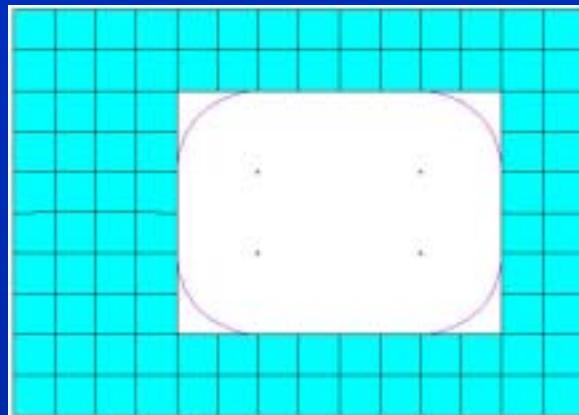
Bonded (Initial contact)

Pilot node can be used together with the bonded contact option to introduce the rigid region between the loading point and the structure, as shown here.

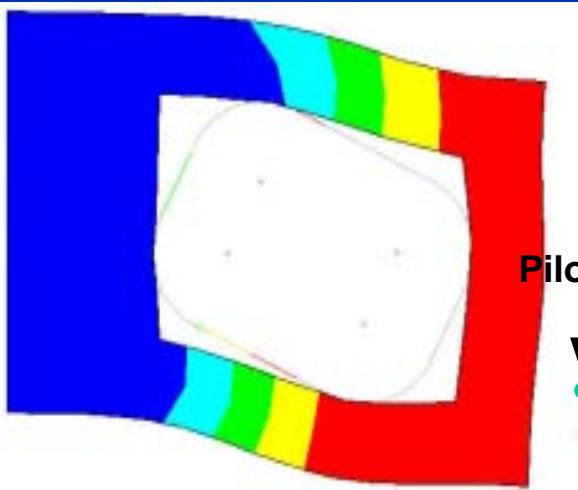
Pilot Node

ANSYS

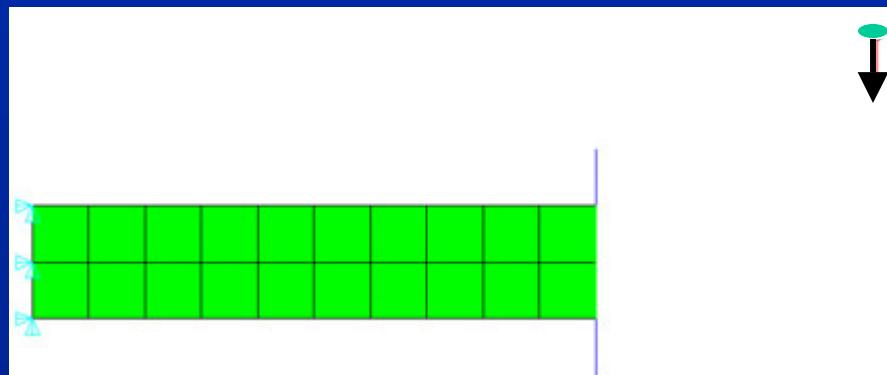
Pilot node:KEYOPT(2)=1



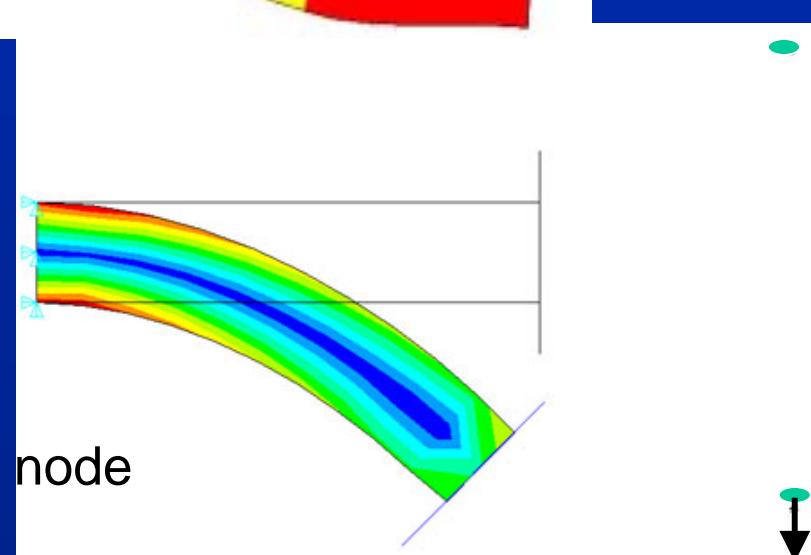
Pilot node



Pilot node



Force FY on Pilot node

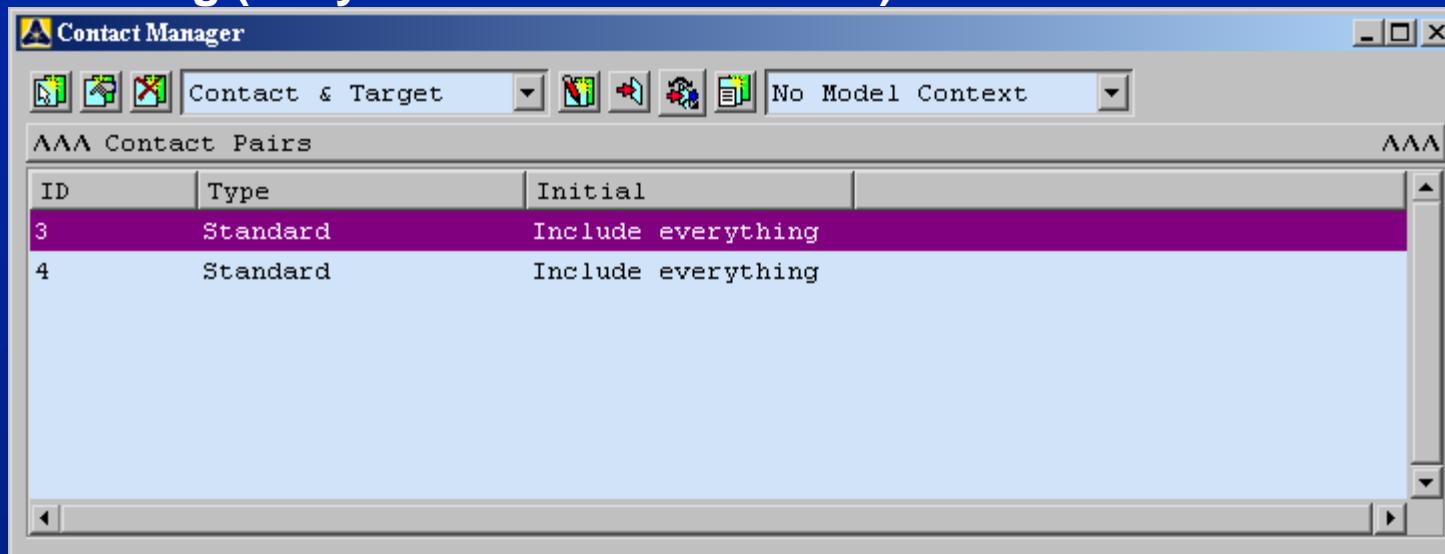


Force FY on Pilot node

Contact Manager

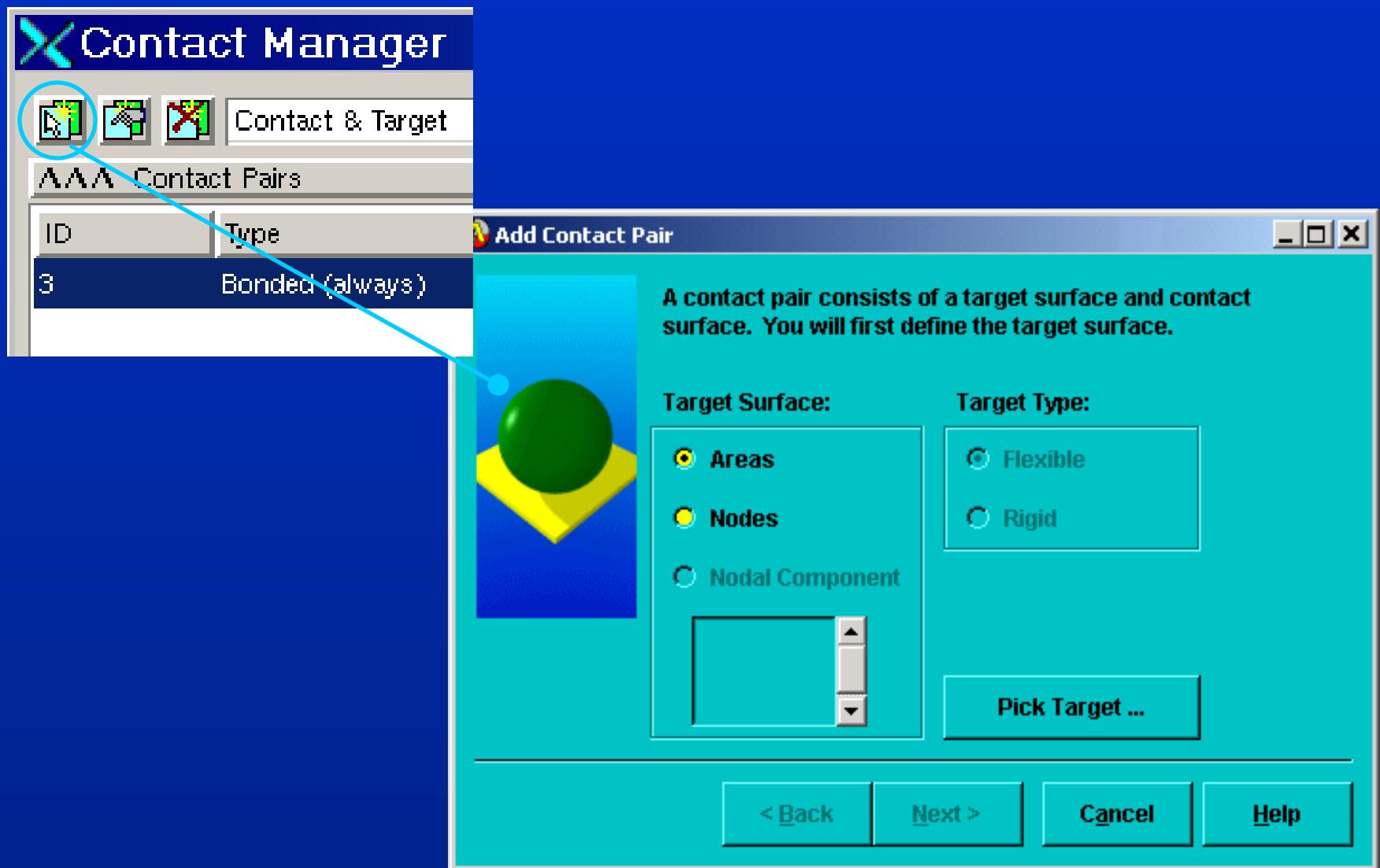


- New Contact Toolbar (allows the user to manage all of their contact definitions in one concise and easy to use viewer.)
 - List contact pairs and some contact properties within a single viewer.
 - Multiple contact pairs can be operated on at the same time for:
 - Modifying Properties (i.e. real constants and keyopts)
 - Plotting Contact Elements (Easy selection and plot)
 - Element Listing (Easy selection and list)
 - Deleting (Easy selection and deletion)



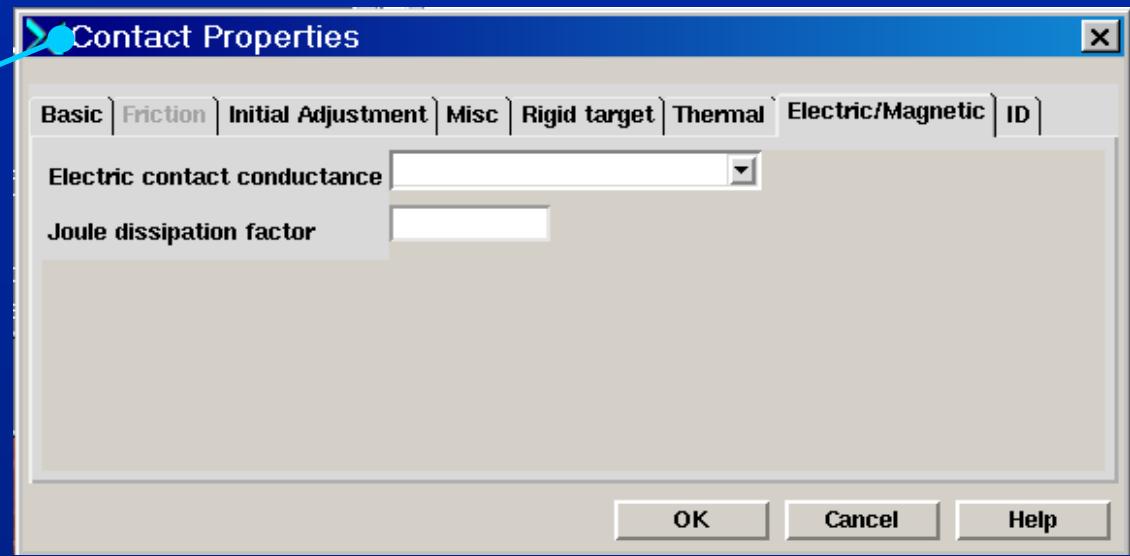
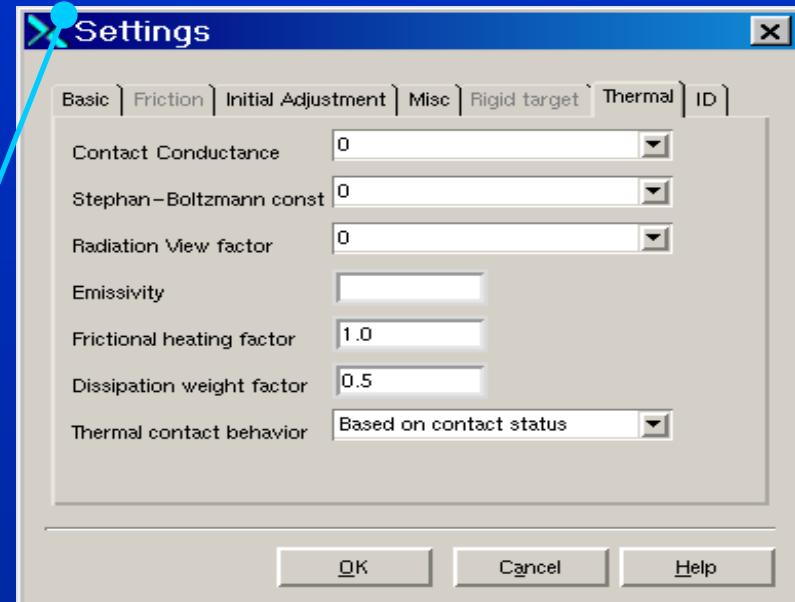
Contact Wizard

ANSYS



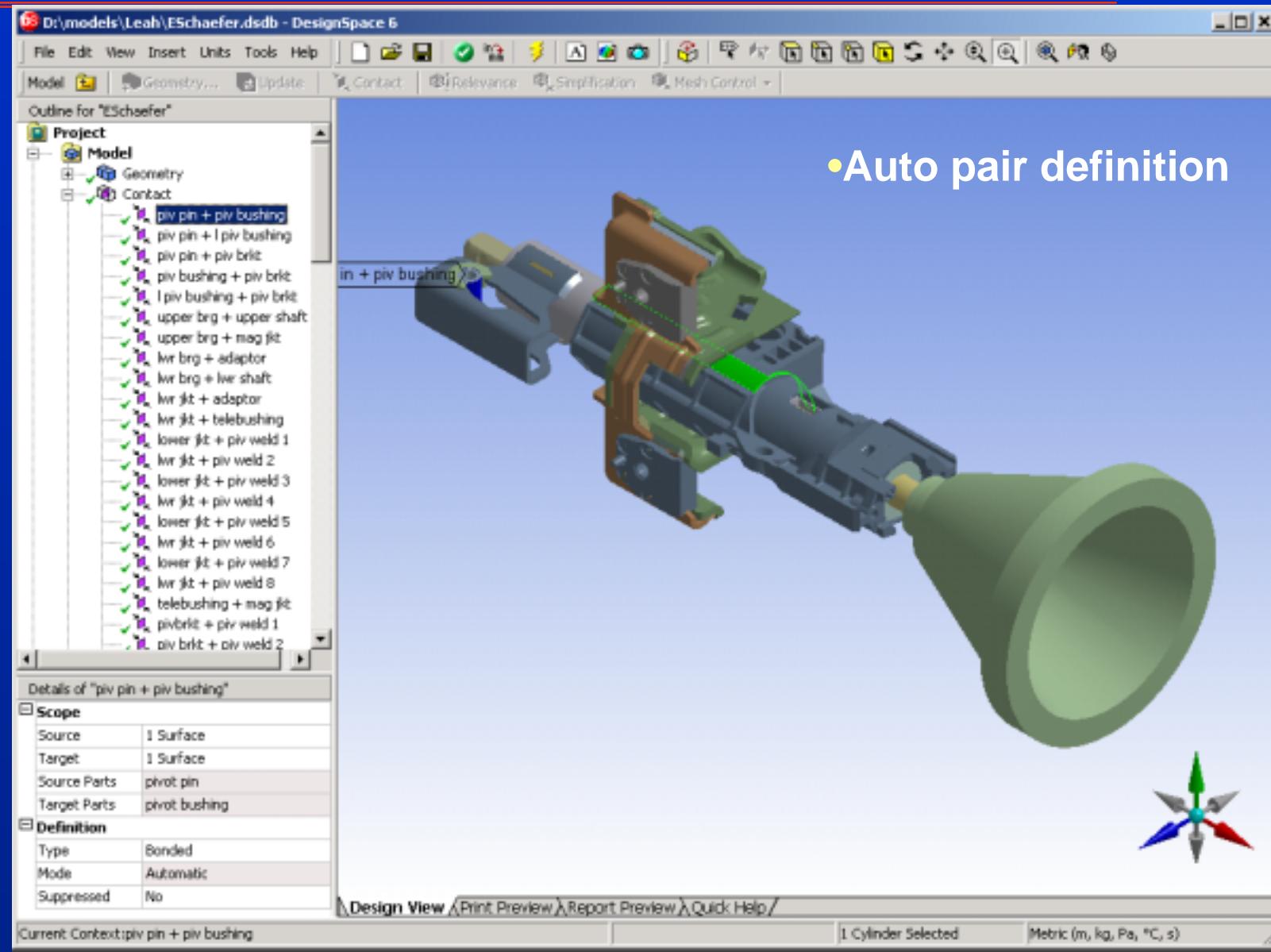
Contact Wizard

ANSYS



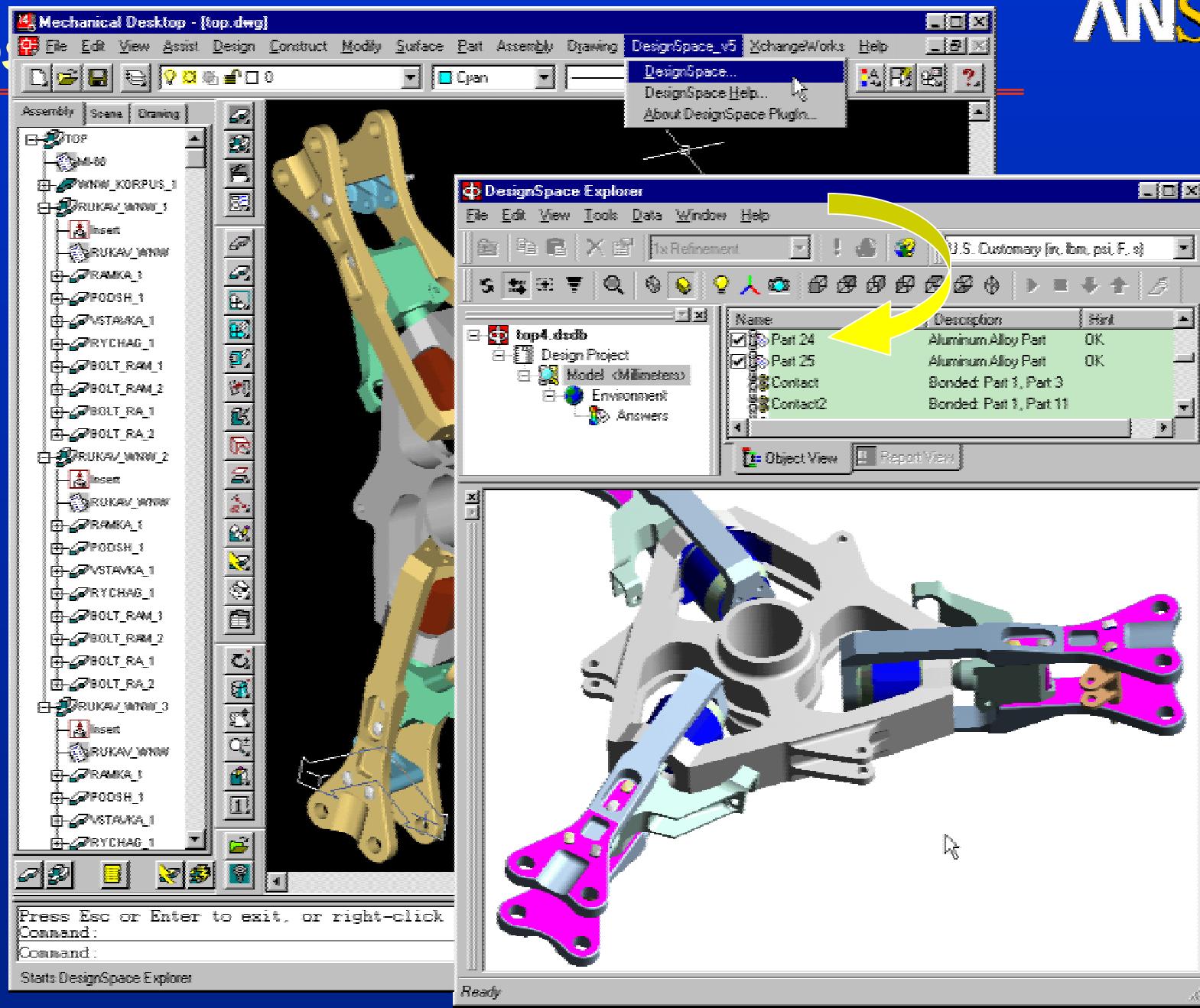
DesignSpace® Graphical User Interface

ANSYS



ANSYS

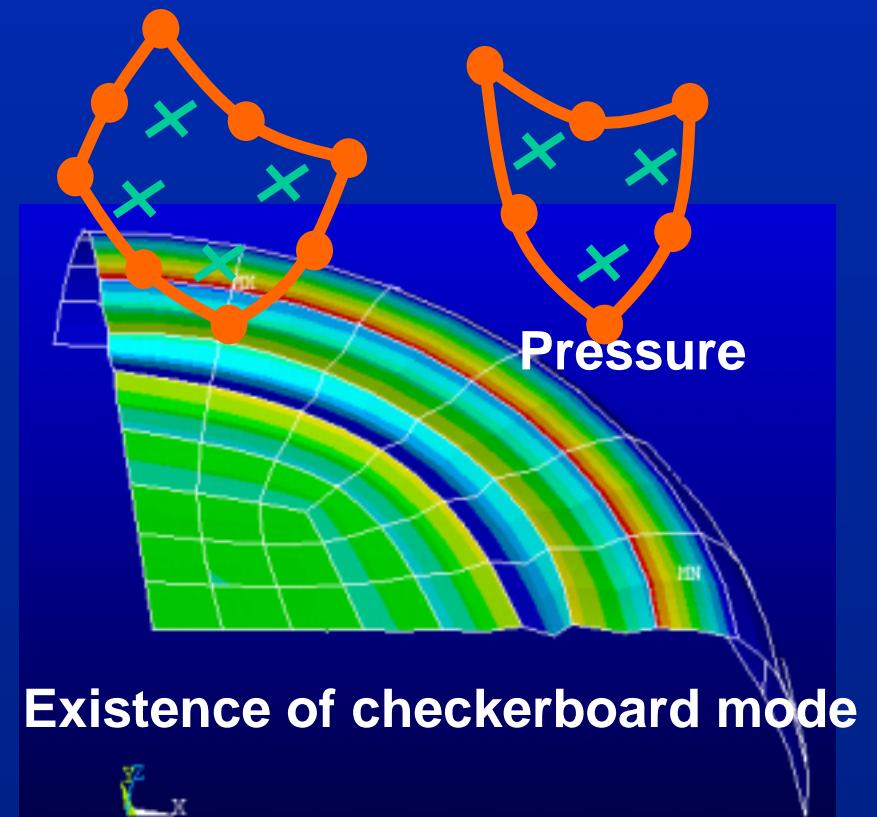
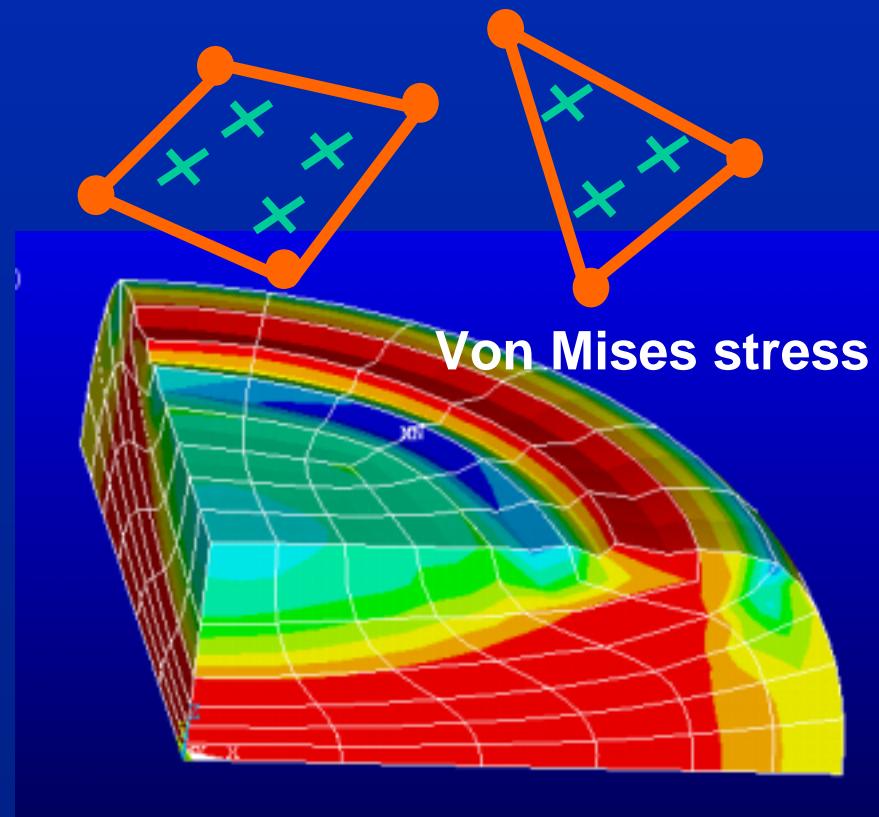
DesignSpace



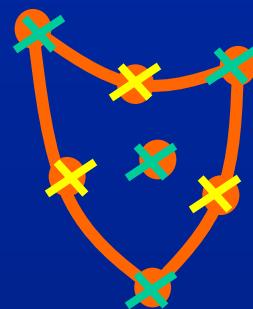
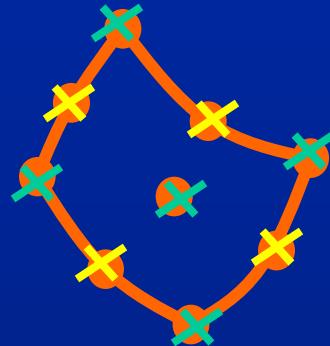
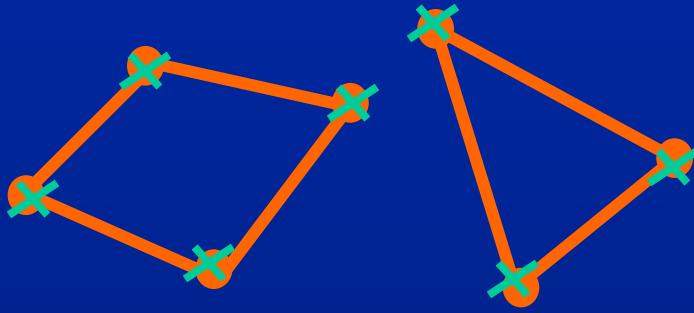
Quadrature Rules

ANSYS

- Gauss full integration rule leads to overconstraining of model (fails LBB condition)
 - Penalty or augmented Lagrange method may alleviate the problem.
 - Smoothing is not required. It passes patch test.



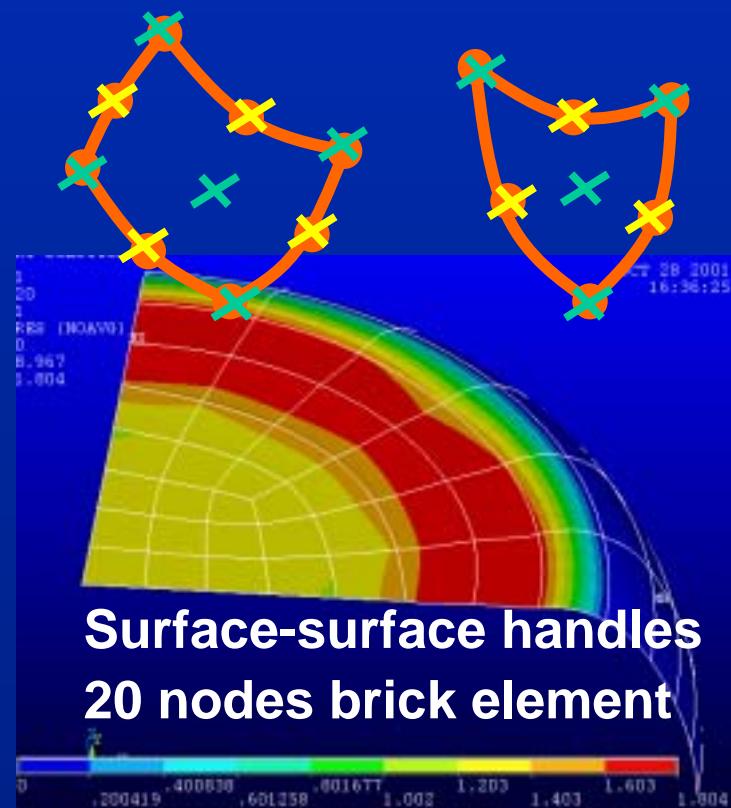
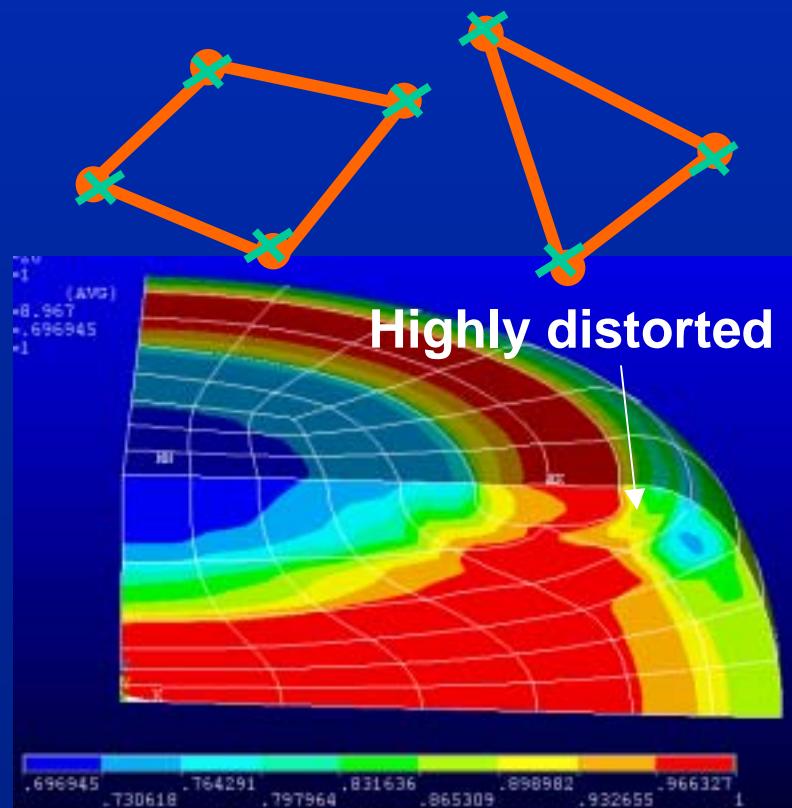
- **Nodal quadrature rule**
 - Smoothing is always required
 - Overconstraint still exists for higher order contact element
 - Add a midface node to the center of contact element to get consistent nodal force
 - Use mixed U-P formation
 - The order of interpolation function for pressure is lower than that for displacement



Quadrature Rules



- Nodal integration rule (satisfies the LBB condition)
 - Smoothing is always required which is performed by averaging surface normals connected to the node
 - Overconstraint exists for quadratic order contact element.
 - Use mixed U-P formation



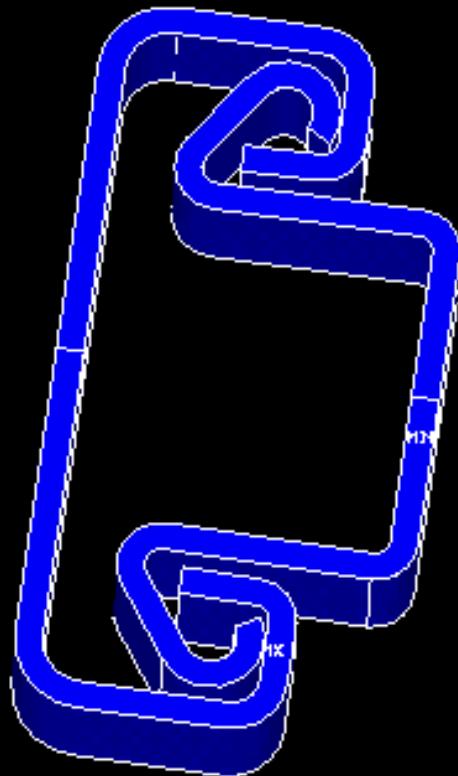
Interconnected Rails

ANSYS

```
1  
NODAL SOLUTION  
STEP=1  
SUB =1  
TIME=.01  
/EXPANDED  
SEQV    (AVG)  
DMX =.07  
SMX =.331E-10
```

ANSYS

FEB 11 2002
18:25:22



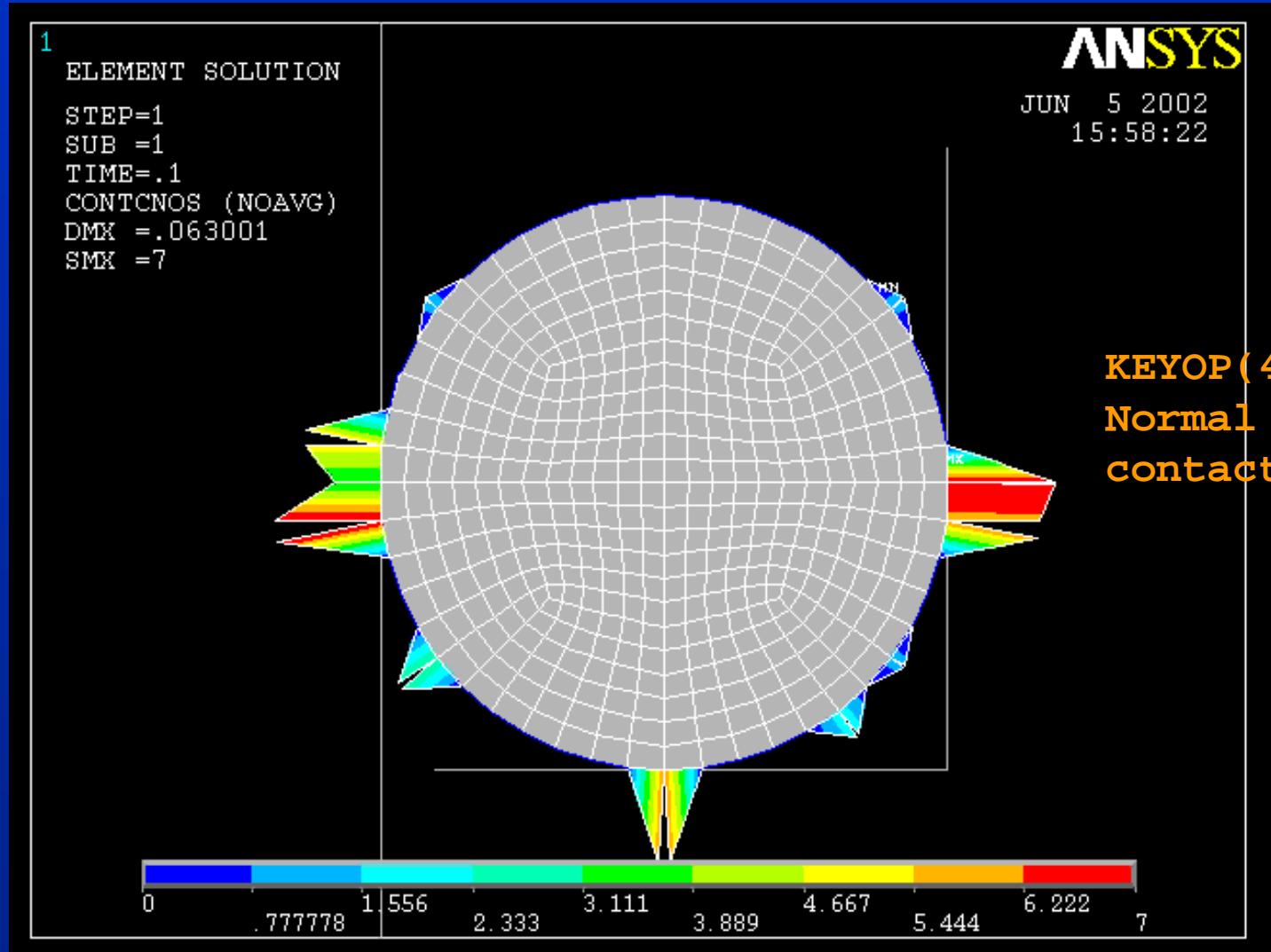
Contact Normal Direction



- KEYOPT(4)=0, Gauss point, perpendicular to contact surface
- KEYOPT(4)=1, Nodal point, perpendicular to contact surface
 - New, smoothing on contact surface is performed which is more expensive than Gauss point Scheme
 - When contact surface is smoother then target surface
- KEYOPT(4)=2, Nodal point, perpendicular to target surface
 - New, smoothing on target surface is performed which is much more expensive than option (KEYOPT(4)=1)
 - When target surface is smoother then contact surface
- All schemes support low/high order contact elements
- Smoothing is always required which is performed by averaging surface normals connected to the node

O-ring Problem

ANSYS



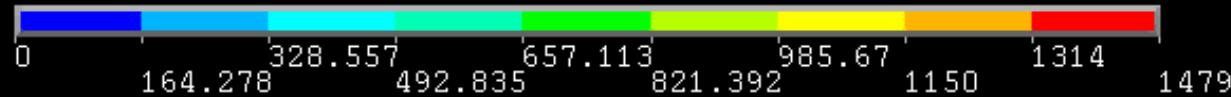
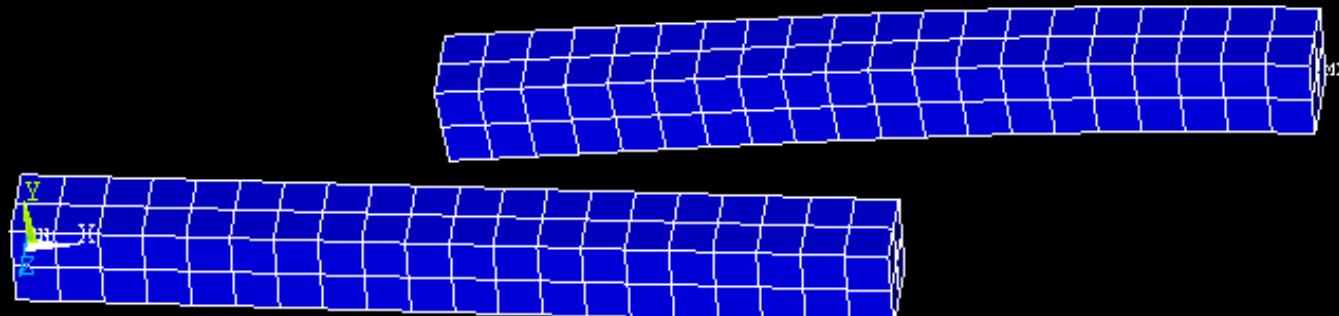
Double Beams Problem

ANSYS

```
1 NODAL SOLUTION  
STEP=1  
SUB =1  
TIME=.1  
SEQV (AVG)  
DMX =6.166  
SMX =130.29
```

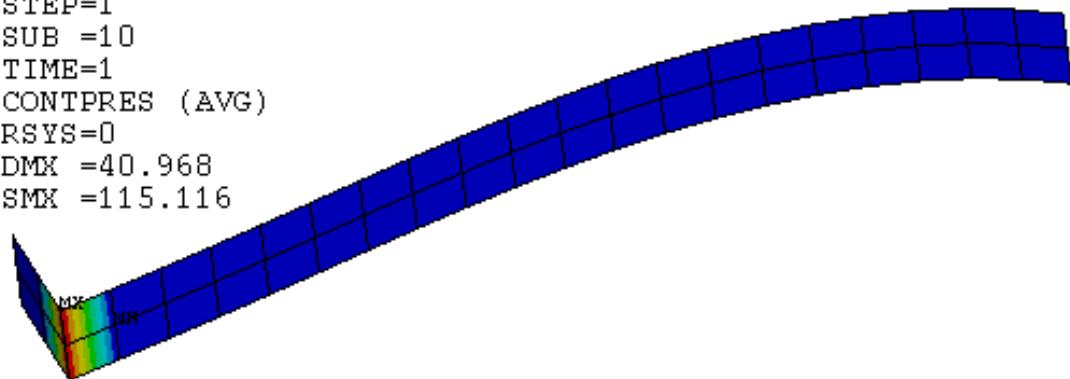
ANSYS
JUN 5 2002
KEYOP(4)=2

Normal to
target surface



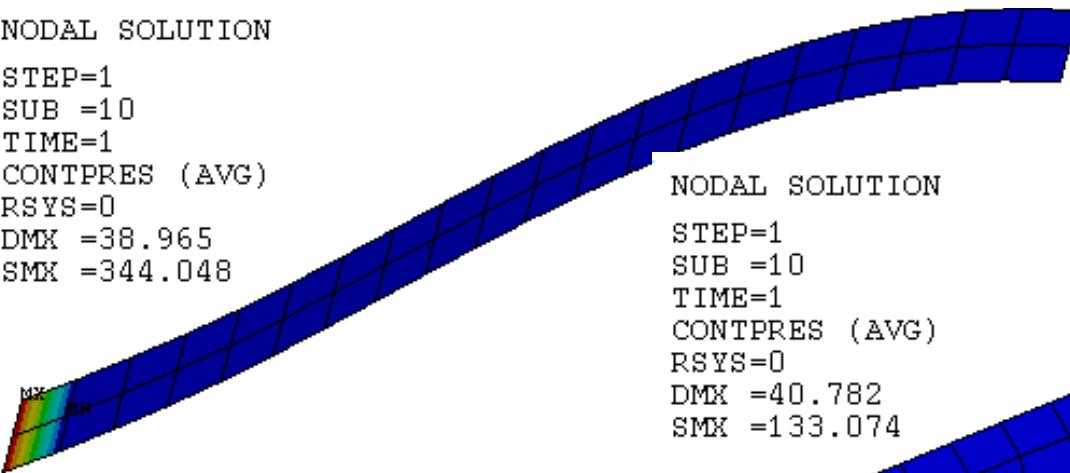
Double Beams Problem

```
STEP=1  
SUB =10  
TIME=1  
CONTPRES (AVG)  
RSYS=0  
DMX =40.968  
SMX =115.116
```



NODAL SOLUTION

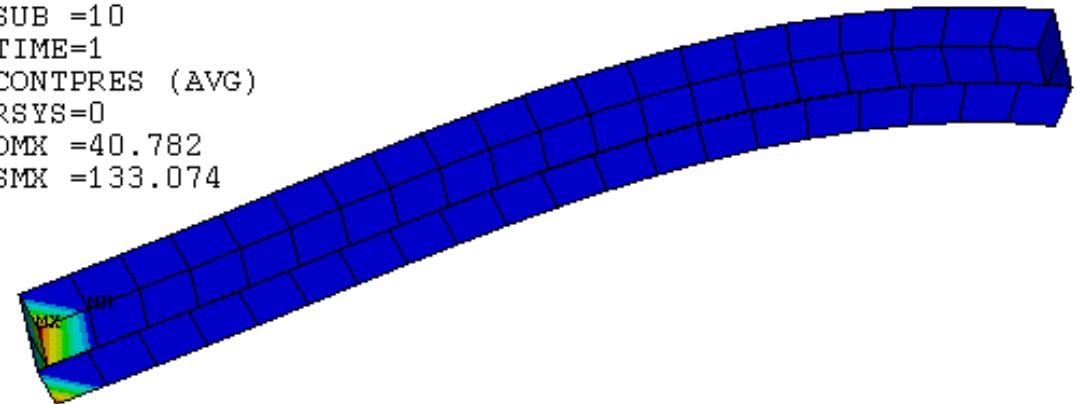
```
STEP=1  
SUB =10  
TIME=1  
CONTPRES (AVG)  
RSYS=0  
DMX =38.965  
SMX =344.048
```



NODAL SOLUTION
STEP=1
SUB =10
TIME=1
CONTPRES (AVG)
RSYS=0
DMX =40.782
SMX =133.074

KEYOP(4)= 1
Normal from contact

Different contact
element pattern gives
different normal
eventually gives
different answers



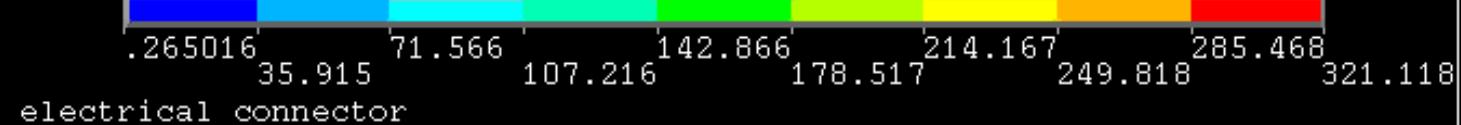
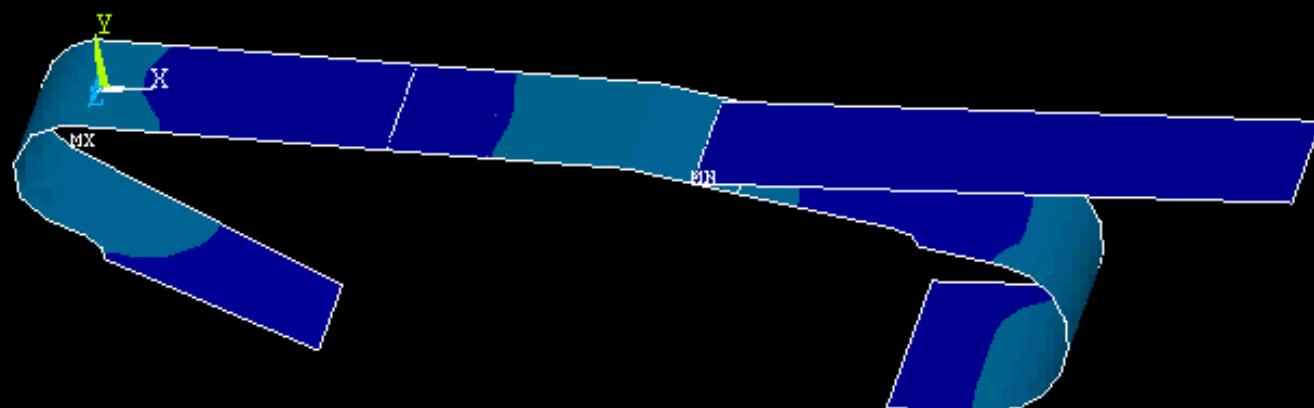
Electrical Connector

ANSYS

```
1  
NODAL SOLUTION  
STEP=1  
SUB =1  
TIME=.05  
SEQV   (AVG)  
DMX  =.15  
SMN  =.265016  
SMX  =74.185
```

ANSYS

FEB 6 2002
16:26:52



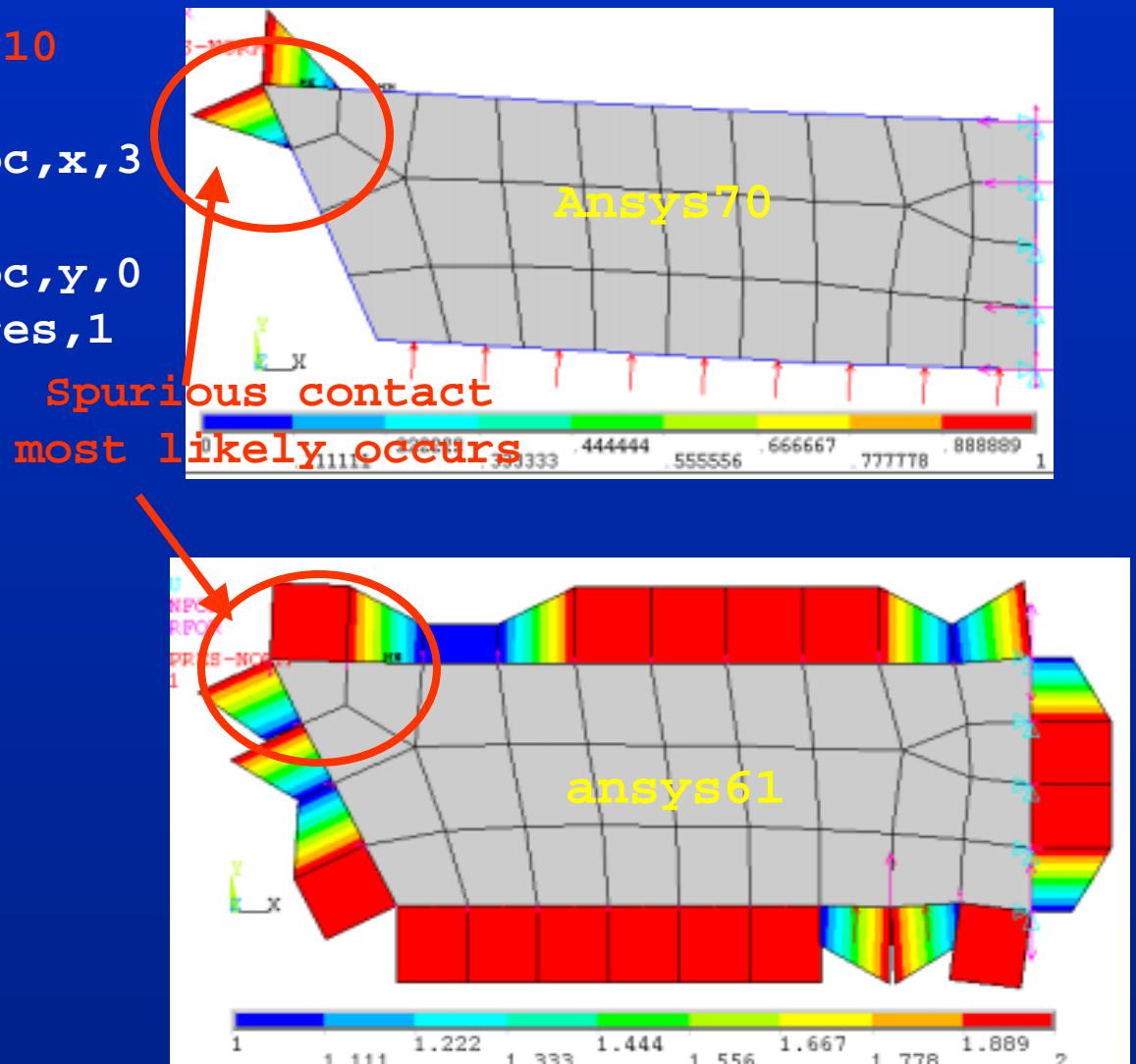
Contact Pair based Setting



- It is contract to individual element based settings.
- Each contact pair uses unique setting:
 - Contact depth, contact length, pinball
 - Normal contact stiffness.
- User can easily monitor and control the settings.
- Contact pressure is proportional to the penetration.
- Mesh size dependence plays less role.
 - Meshes with large variations in element sizes.
- It often results in better convergence.

Auto Spurious Prevention: KEYOP(8)=1 is no long needed

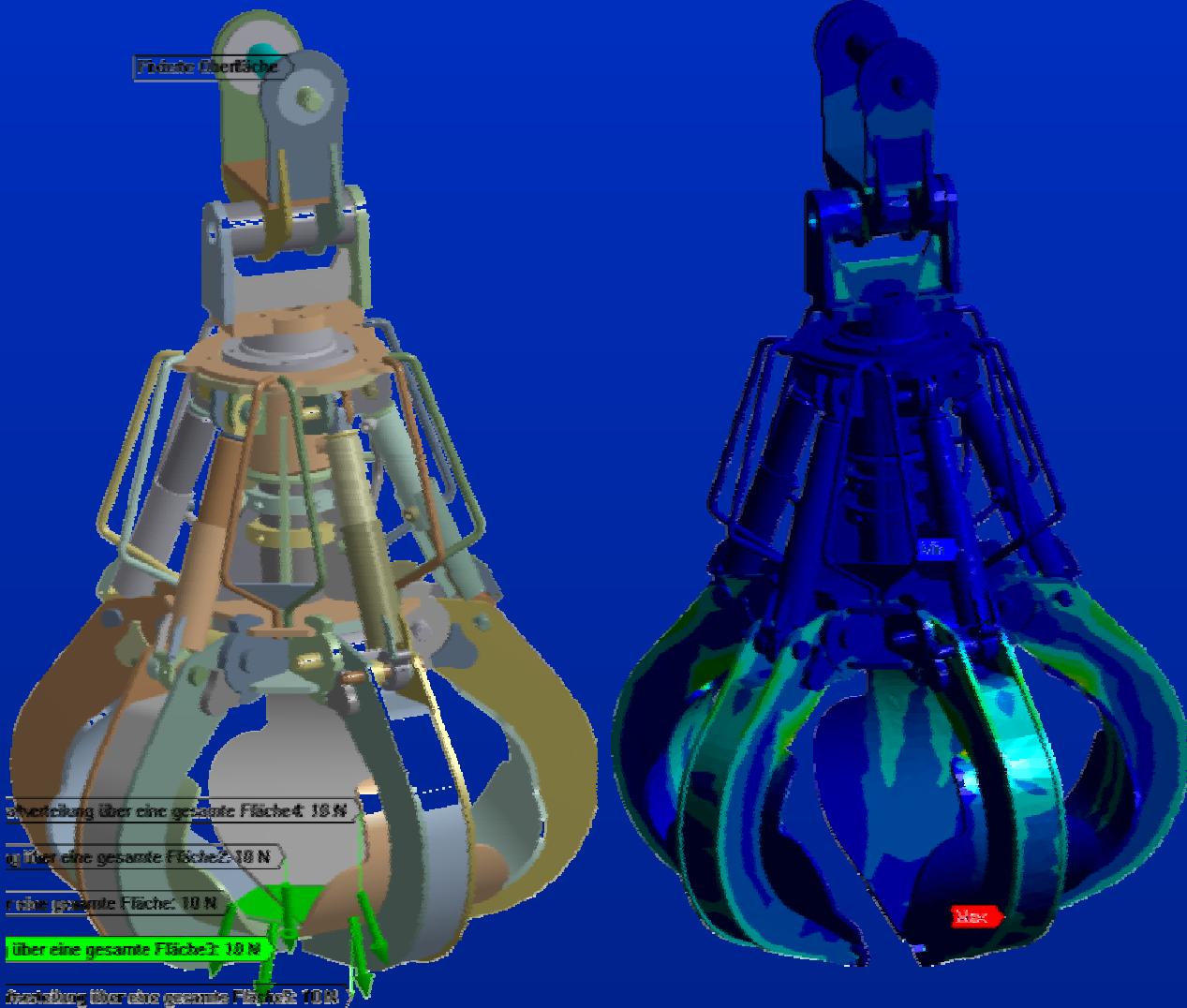
```
/batch,list rmod,1,6,10
/prep7 /solu
et,1,182,2 nsel,s,loc,x,3
et,2,169 d,all,all
et,3,175 nsel,s,loc,y,0
keyop,3,4,1 sf,all,pres,1
keyopt,3,8,0 alls
mp,ex,1,2e5 solv
rect,0,3,0,1
kmod,1,0.5
esiz,0.3
ames,all
type,3
esurf
type,2
esurf
```



- Ansys70 can automatically detect asymmetric contact pair when symmetric pairs are defined
 - Many pairs exists in the model
 - Be difficult to pick
 - Model came from DS using auto contact detection
 - The distinction between the contact and target surfaces is not clear.
 - The contact results are very hard to be interpreted.
 - Overstrained model
 - Decision is made by many factors
 - Stiffness, numb of element, element size, area, curvature

Believe or not

ANSYS



It has 209 unsuppressed parts and 450 contact pairs. The model has 385,000 nodes and 1.15million DOF

Automatic Asymmetric Detection



*** NOTE ***

CP= 19.490 TIME= 18:22:37

Symmetric Deformable- deformable contact pair identified by real constant set 8 and contact element type 9 has been set up. The companion pair has real constant set ID 9. Both pairs should have the same behavior.

ANSYS will keep the current pair and remove its companion pair, resulting in asymmetric contact.

*** NOTE ***

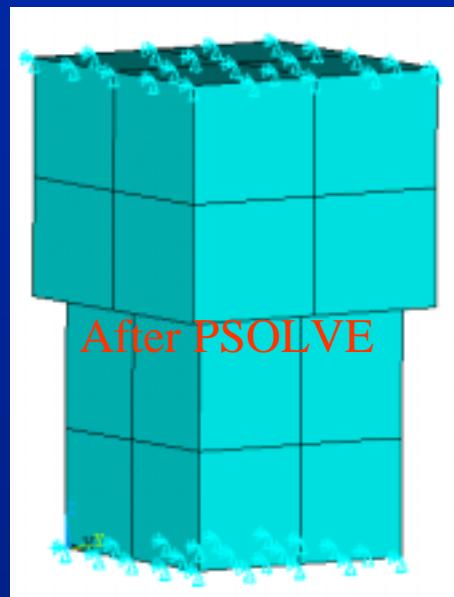
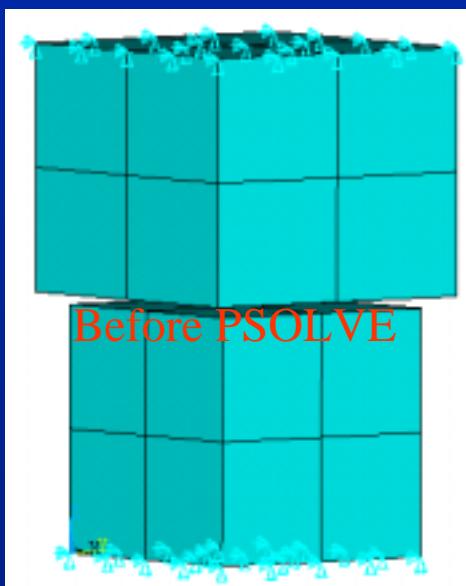
CP= 19.490 TIME= 18:22:37

Symmetric Deformable- deformable contact pair identified by real constant set 9 and contact element type 9 has been set up. The companion pair has real constant set ID 8. Both pairs should have the same behavior.

ANSYS will remove the current pair and keep its companion pair, resulting in asymmetric contact.

Initial Contact Adjustment

- PSOLVE, ELFORM
 - Check initial contact status, db -> rdb
 - Physically moves contact nodes to the target surface
 - Only for nodal detection option or CONTA175
 - Initial contact nodes inside ICONT zone
 - Initial penetrated nodes with KEYOP(9)=1
 - After save, all the setting can be modified



Output (if CONCNTR,PRINT,>0)

Node 22 moved to target surface

Node 25 moved to target surface

Contact Algorithms



- Penalty method (**KEYOP(2)=1**)
- Augmented Lagrange method (**KEYOP(2)=0**)
- Lagrange multiplier method (**KEYOP(2)=3**)
- Lagrange multiplier on normal & penalty on tangent (**KEYOP(2)=2**)
- Automatic adaptation of boundary conditions and constraints
- Each method has advantages and disadvantages depending on the particular problem considered

Penalty Method



- Some penetration occurs with the amount being determined by the penalty constants.
- The total potential energy (virtual work)

$$\delta\Psi = \int_V \sigma^T \delta\varepsilon dV + \int_{\Gamma} (\varepsilon_N g_N \delta g_N + \varepsilon_T \mathbf{g}_T \delta \mathbf{g}_T) dA$$

- $\varepsilon_T \mathbf{g}_T \delta \mathbf{g}_T$ is associated with tangential sticking
- $\varepsilon_T \mathbf{g}_T \delta \mathbf{g}_T \rightarrow \mathbf{t}_T \delta \mathbf{g}_T$ for sliding, the frictional stress is determined by the frictional law
- The Lagrange multiplier method can be recovered if
 - $\varepsilon_N \rightarrow \infty; \varepsilon_T \rightarrow \infty$
- However this will lead to ill-conditioning problem

Penalty Method



- The linearized form :

$$-\mathbf{u}_{old} = \bar{\mathbf{K}}\delta\mathbf{f} + \mathbf{B}^T\delta\mathbf{p} + \delta\mathbf{B}^T\mathbf{p} = \bar{\mathbf{K}}\delta\mathbf{f} + \mathbf{K}_\sigma(p, t)\delta\mathbf{f} + \mathbf{B}^T\mathbf{C}\mathbf{B}\delta\mathbf{f}$$

- The system equation in New-Raphson iteration form

$$\left[\bar{\mathbf{K}} + \mathbf{K}_\sigma(\mathbf{\varepsilon}_N g_N, \mathbf{\varepsilon}_T \mathbf{g}_T) + \mathbf{B}^T \mathbf{C} \mathbf{B} \right] \delta\mathbf{f} = -\mathbf{u}_{old}$$

- Advantages: It is simple and displacement based FE framework remains
- Disadvantages: It suffers from ill-conditioning as penalty stiffnesses are increased.

Augmented Lagrange Method



- Ill-conditioning can be alleviated by augmenting Lagrange multipliers
- The total potential energy (virtual work)

$$\delta\Psi = \int_{\Gamma} [(\lambda_N + \varepsilon_N g_N) \delta g_N + (\lambda_T + \varepsilon_T \mathbf{g}_T) \delta \mathbf{g}_T] dA$$

- It can be considered as a generalization of the Lagrange multiplier method where an additional term involving the contact tractions is added to the variational equations.
- Augmented Lagrange method at element level
 - If the Lagrange multipliers are retained as variables in the active contact elements and they remain as element level variables and do not enter the global structural solution.

Augmented Lagrange Method

ANSYS

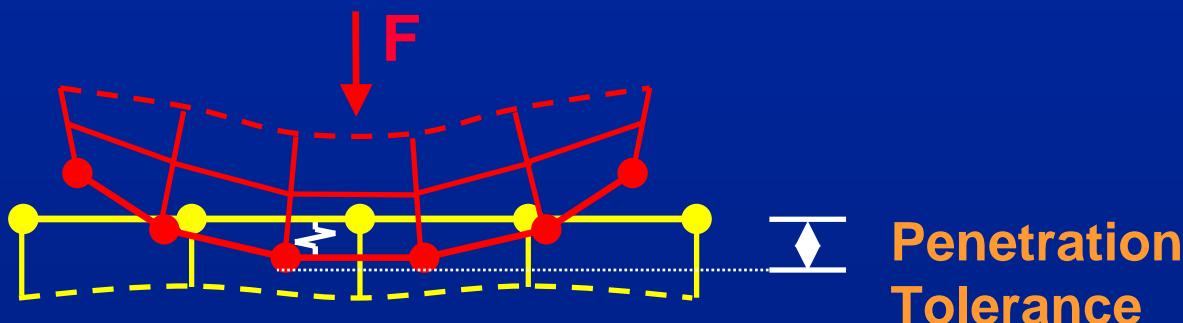
- The resulting New-Raphson form:

$$[\bar{\mathbf{K}} + \mathbf{K}_\sigma(\lambda_N + \varepsilon_N g_N, \lambda_T + \varepsilon_T \mathbf{g}_T) + \mathbf{B}^T \mathbf{C} \mathbf{B}] \{\delta \mathbf{f}\} = -\{\mathbf{u}_{old}\}$$

- The penalty stiffnesses need not be very large because the contact constraint is satisfied via the Lagrange multipliers augmented during iteration:

$$\lambda = \lambda_{old} + \mathbf{C} \mathbf{g}$$

- Check $|g_N| < \text{tolerance}$
- Penetration control with additional iterations
- Less sensitivity to penalty stiffnesses



Augmented Lagrange Method



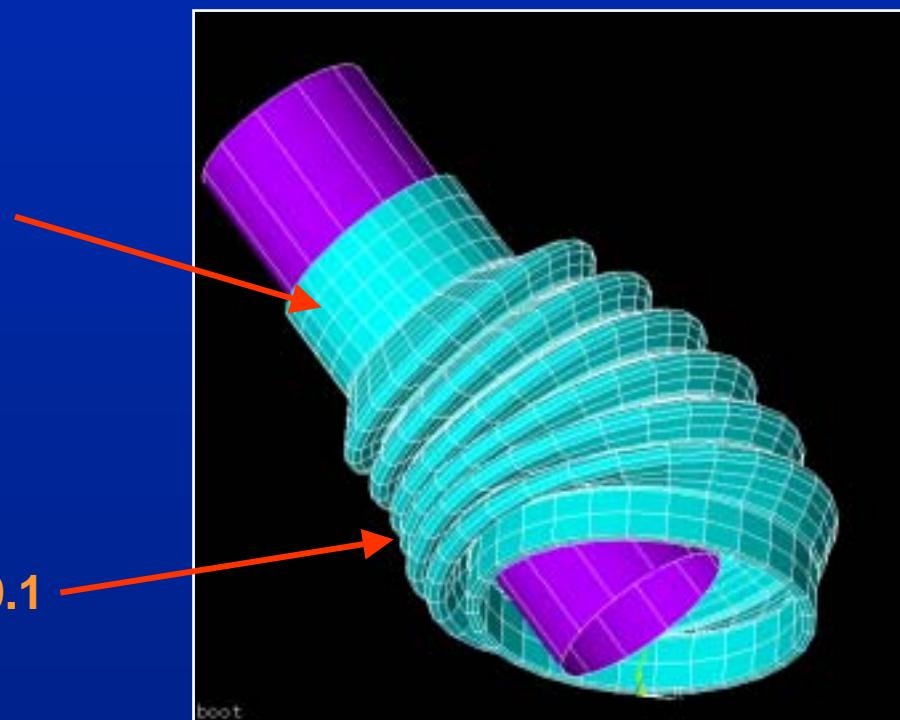
- **Penalty stiffness is the most important parameter affecting accuracy and convergence behavior**
 - High stiffness
 - Less penetration (better accuracy)
 - ill-conditioning (more difficult convergence)
 - Low stiffness
 - May help convergence, but lead to more penetration
 - Need more iterations if using too small penetration tolerance
- **What's default for normal contact stiffness**
 - Bulk modulus & material behaviors of underlying elements
 - Size of element (depth), structural flexibilities, self-contact
 - Total DOF
 - Cover 80-90 % applications

Contact Stiffness

ANSYS

- Normal penalty coefficient FKN
 - 1.0 (default) for bulk solid in contact
 - 0.1 for more flexible (bending-dominated) parts
- Default tangent penalty coefficient FKT
 - Can be overwritten

Bulky contact; try FKN = 1.0



Flexible contact; try FKN = 0.1

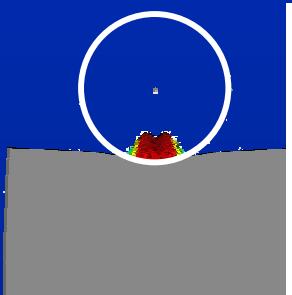
Penetration VS. Contact Pressure

ANSYS

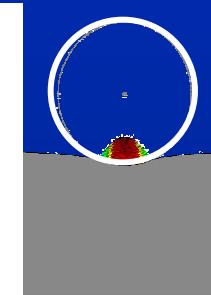
d is result from FKN and equilibrium equation

Pressure = $d * \text{FKN}$ => Contact stress on contact surface

100-times Difference with FKN leads to 100-times Difference with d
but only leads to about 5% Difference with contact pressure and stress.

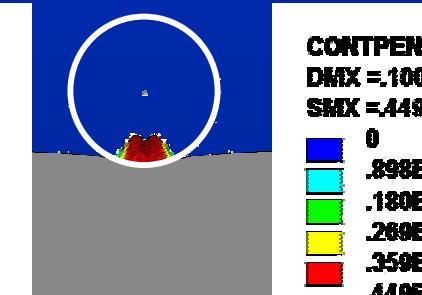


CONT PENE (A)	
DMX	= .1
SMX	= .259E-03
0	
.518E-04	
.104E-03	
.155E-03	
.207E-03	
.259E-03	



CONT PRES (A)	
DMX	= .1
SMX	= 43642
0	
8728	
17457	
26185	
34914	
43642	

Penetration $d=0.26e-3, P=43642$



CONT PENE (A)	
DMX	= .10008
SMX	= .449E-05
0	
.898E-06	
.180E-05	
.260E-05	
.350E-05	
.449E-05	

Penetration $d=0.45e-5, P=43927$



CONT PRES (A)	
DMX	= .10008
SMX	= 43927
0	
8785	
17571	
26356	
35142	
43927	

$$\Delta d = \frac{0.26e-3}{0.45e-5} = 60$$

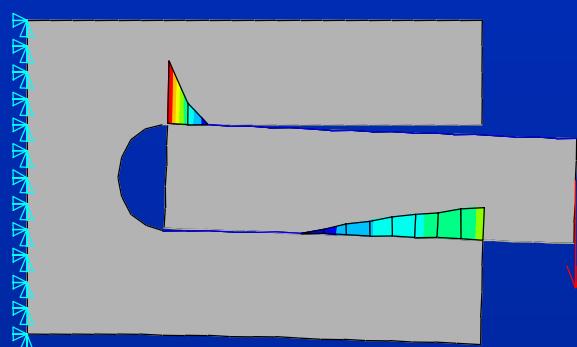
$$\Delta p = \frac{43927}{43642} = 1.007$$

Local Accuracy VS. Global Equilibrium

ANSYS

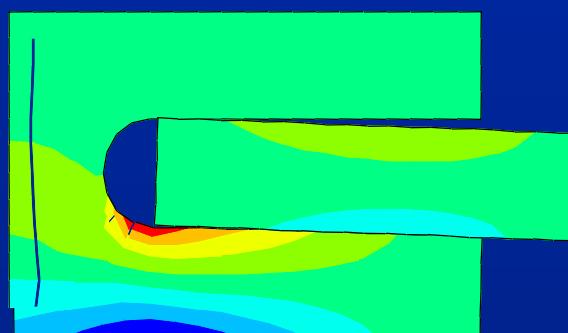
KN=0.001 und TOLN=0.1

Penetration



0
.002202
.004404
.006606
.008808
.01101
.013212
.015414
.017616

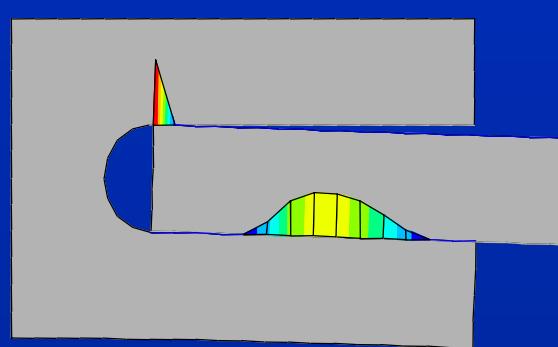
Bending stress



-123.031
-88.59
-54.148
-19.707
14.735
49.176
83.618
118.059
152.501

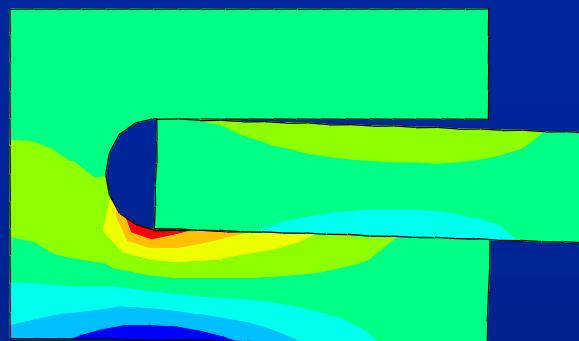
KN=0.001 und TOLN=0.01

Penetration



0
.432E-03
.864E-03
.001296
.001728
.002159
.002591
.003023
.003455

Bending stress



-121.92
-87.795
-53.671
-19.547
14.578
48.702
82.826
116.951
151.075

Contact Stiffness Update



- KEYOP(10)=0 – update contact stiffnesses at each load step if FKN, FKT are redefined
- KEYOP(10)=1 – update contact stiffnesses at each sub-step based on current geometry
- KEYOP(10)=2 – update contact stiffnesses at each iteration – adaptive scheme

Contact Stiffness Update



- Adaptive scheme for normal contact stiffness
 - First iteration bases on previous convergence pattern

$$\boldsymbol{\varepsilon}_N^1 = \begin{cases} 0.2 \bullet \boldsymbol{\varepsilon}_N^0 & \text{if } k > 10 \quad \text{or} \quad \text{diverge} \\ \boldsymbol{\varepsilon}_N^0 & \text{if } k \leq 10 \end{cases}$$

- The subsequent iterations depend on penetration

$$\boldsymbol{\varepsilon}_N^{K+1} = \begin{cases} 5 \bullet \boldsymbol{\varepsilon}_N^K & \text{if } g_N^{K+1} > \frac{1}{4} g_N^K \\ \boldsymbol{\varepsilon}_N^K & \text{if } g_N^{K+1} \leq \frac{1}{4} g_N^K \end{cases}$$

Contact Stiffness Update



- Adaptive scheme for tangential stiffness
 - Based on current normal pressure

$$\varepsilon_T = \frac{\mu p}{l_{crit}} \quad l_{crit} \quad \text{Allowable Max. elastic slip} = 0.01 * l$$

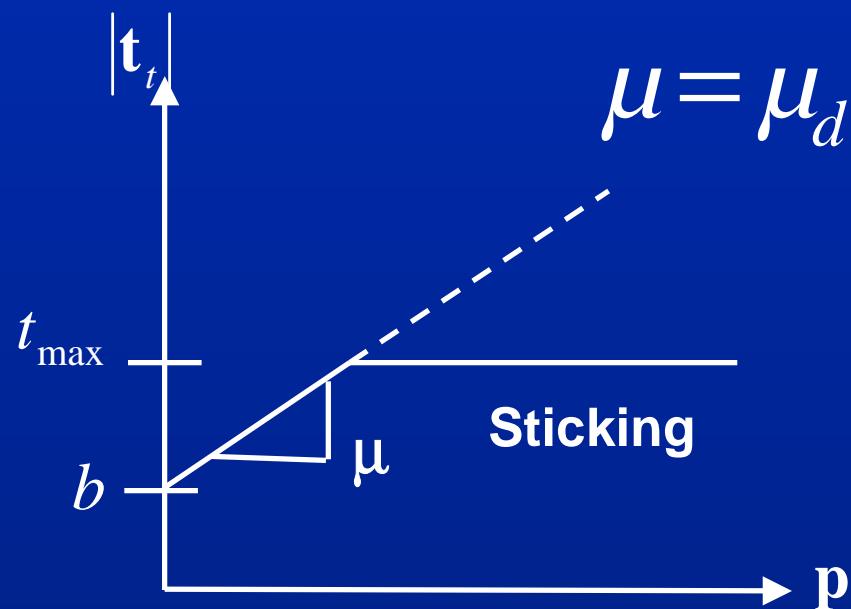
- Allowable Max. elastic slip distance (SLTOL)
- Positive: as a factor \rightarrow SLTOL*(contact length)
- Negative: as a true value
- Default to 1.d0

Friction Overview



- Friction is a complex physical phenomena that involves the characteristics of the surface
 - Surface roughness, temperature, pressure, and relative velocity
- Friction is a path dependent process, certain history variables must be tracked and accounted for traction determination.
- The definition of reference system is a key point for integrations of the derivatives of contact tractions.

- Coulomb model with additional cohesion b and shear behavior t_{\max}
 - Sticking contact is reversible
 - Sliding contact is irreversible
- Dynamic friction model



$$\mu = \mu_d + (\mu_s - \mu_d) e^{-d_c g}$$

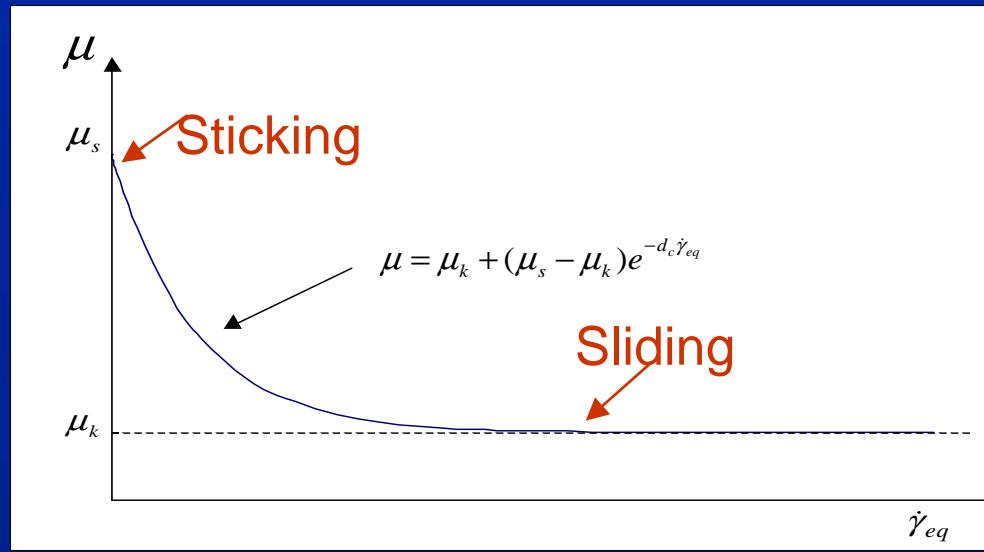
•
Static, dynamic, decay
friction coefficients

Friction Law

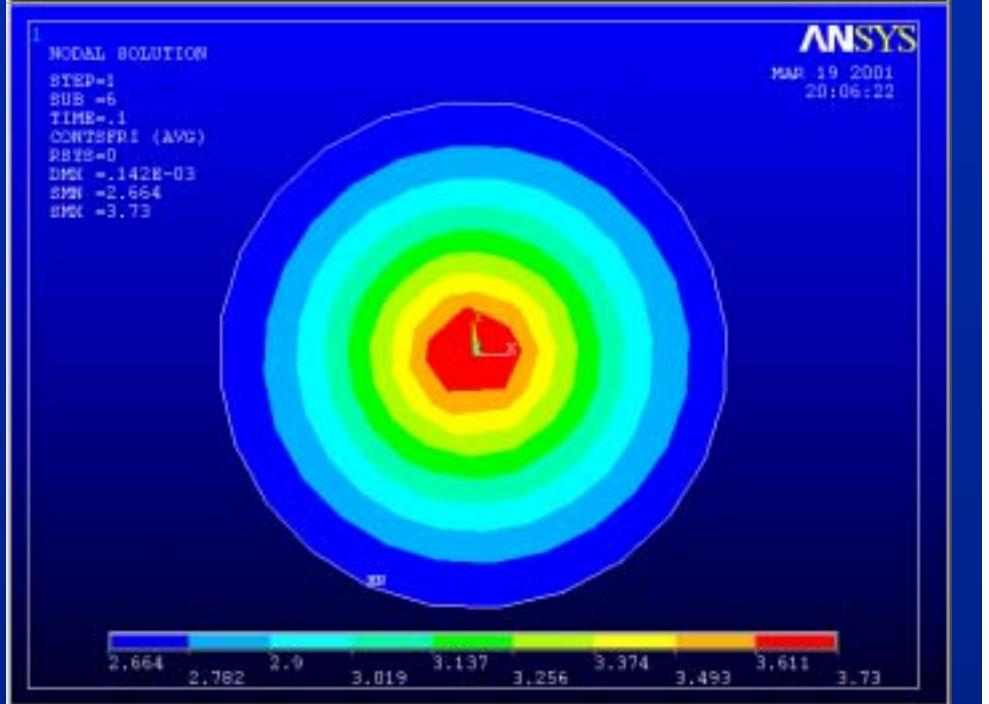
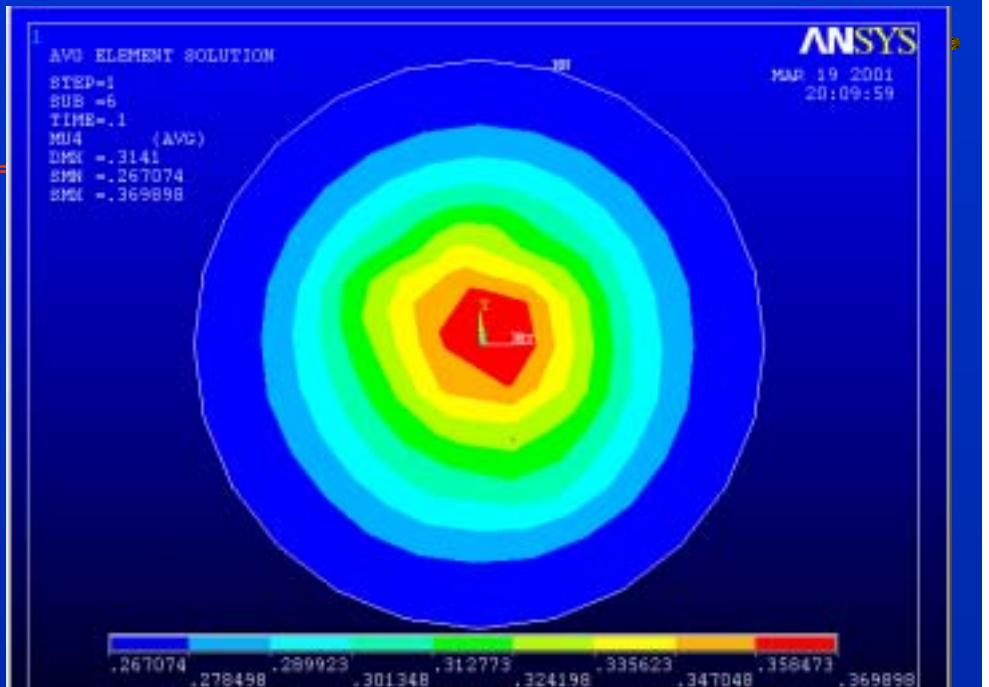
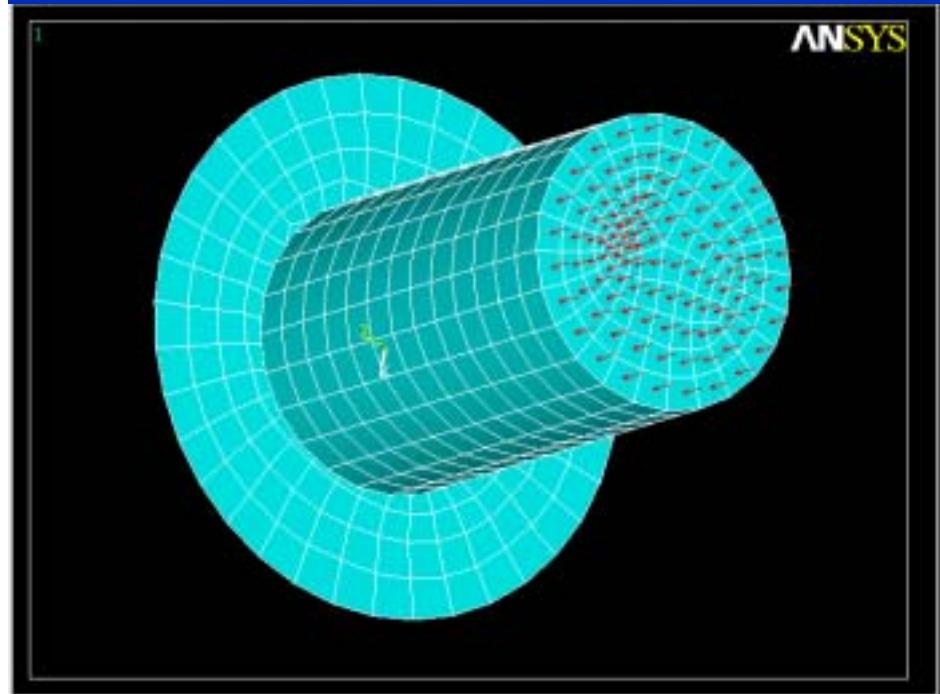
ANSYS

- If the static and dynamic coefficients of friction at least one data point (μ_1 ; V_{rel1}) are known
- The decay coefficient:

$$d_c = -\frac{1}{V_{rel1}} * \ln \left(\frac{\mu_1 - \mu_k}{\mu_s - \mu_k} \right)$$

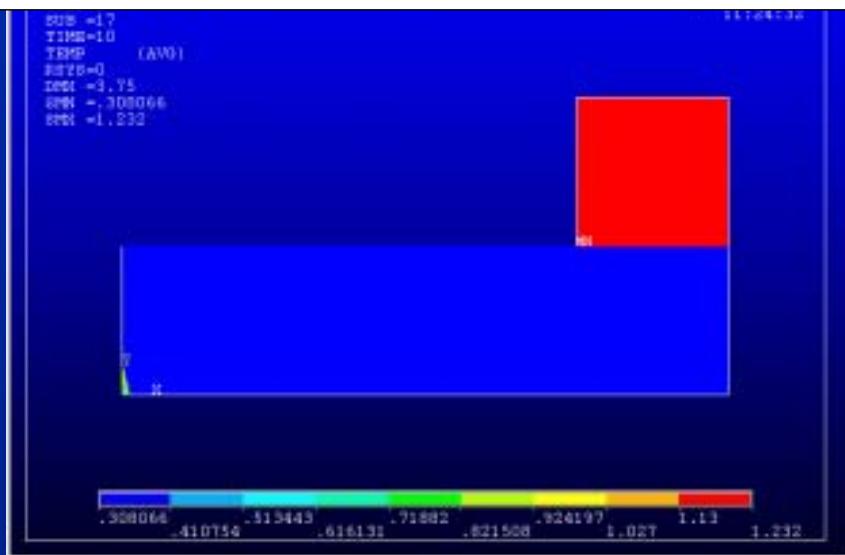
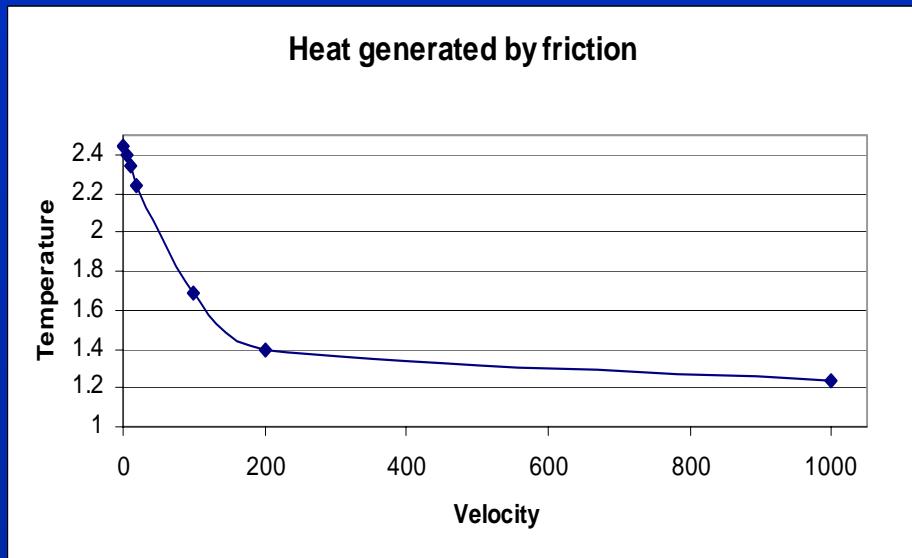
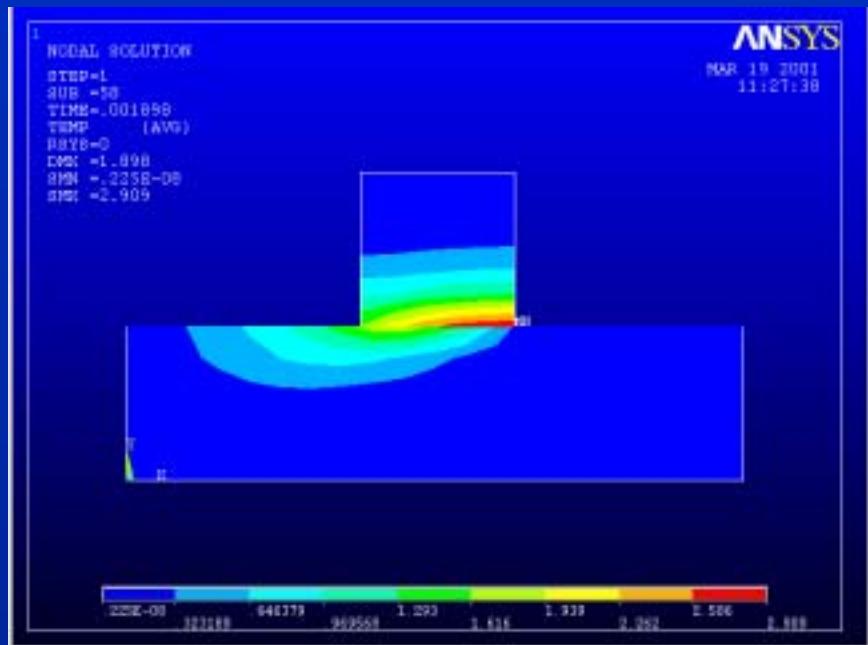


Dynamic Friction



Dynamic Friction

ANSYS



Friction Conditions



- Kuhn-Tucker conditions for Coulomb friction

$$\Phi \equiv |\mathbf{t}_T| - b - \mu p \leq 0 \quad \text{Coulomb friction condition}$$

$$\dot{\mathbf{u}}_T - \xi \frac{\partial}{\mathbf{t}_T} \Phi = \frac{1}{\varepsilon_T} \left(\dot{\mathbf{t}}_T - \dot{\lambda}_T \right) \quad \text{Non-associated flow rule}$$

Tangential traction is applied in the sliding direction

$$\xi \geq 0$$

Sliding only occurs when $\Phi=0$

$$\xi \Phi = 0$$

If $\Phi < 0$ sticking occurs

Integration of Friction Law



- Update of the tangential traction due to friction is performed by the Return mapping algorithm

$$\mathbf{t}_T^{k+1} = \begin{cases} \mathbf{t}_T^k + \boldsymbol{\varepsilon}_T \Delta \mathbf{g}_T^k & \text{if } |\mathbf{t}_T^n + \boldsymbol{\varepsilon}_T \Delta \mathbf{g}_T^k| - b \leq \mu p^{k+1} \\ t_{\text{crit}}^{k+1} \mathbf{n}_T^{k+1} & \text{otherwise} \end{cases}$$

$$t_{\text{crit}}^{k+1} = \begin{cases} \mu p^{k+1} + b & \text{if } \mu p^{k+1} + b < t_{\text{max}} \\ t_{\text{max}} & \text{otherwise} \end{cases}$$

$$\mathbf{n}_T^{k+1} = \mathbf{n}_T^{tr,k+1} = \frac{\mathbf{t}_T^{tr}}{|\mathbf{t}_T^{tr}|} = \frac{\mathbf{t}_T^k + \boldsymbol{\varepsilon}_T \Delta \mathbf{g}_T^k}{|\mathbf{t}_T^k + \boldsymbol{\varepsilon}_T \Delta \mathbf{g}_T^k|}$$

Integration of Friction Law



$$dt_i = \epsilon_T \frac{t_{crit}^{k+1}}{\left| \mathbf{t}_T^{tr} \right|} (\delta_{ij} - n_i n_j) dg_j \quad \text{if } t_{crit}^{k+1} = t_{max}$$

$$dt_i = \epsilon_T \frac{t_{crit}^{k+1}}{\left| \mathbf{t}_T^{tr} \right|} (\delta_{ij} - n_i n_j) dg_j + \left(\mu + p^{k+1} \frac{\partial \mu}{\partial p} \right) n_i dp \quad \text{if } t_{crit}^{k+1} = \mu p^{k+1} + b$$

$$dt_i = \epsilon_T dg_i + \frac{\Delta g_i}{l_{crit}} \mu dp \quad \text{If sticking}$$

- The matrix is unsymmetric which corresponds to the non-associativity of Coulomb's frictional law
- The combination of the frictional interface law with return mapping algorithm leads to a consistent matrix.

Consistent Stiffness Matrix

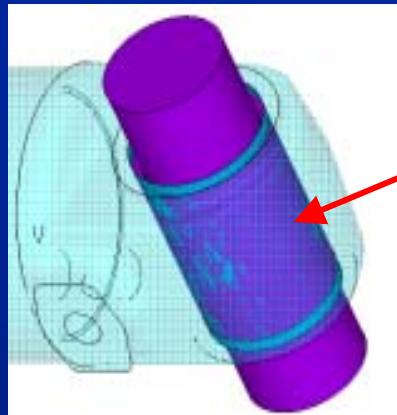


- **Consistent behavior**
 - It makes system equation converge quadratically
- **Unsymmetric behavior**
 - Frictional sliding contact
 - Stress stiffness matrix due to curved contact/target surfaces
 - Adaptive tangential contact stiffness
 - Use the unsymmetric solver even friction coefficient is small
- **Stiffness matrix symmetrization**
 - Maintain limit pressure if sliding
 - Efficient for most contact problems

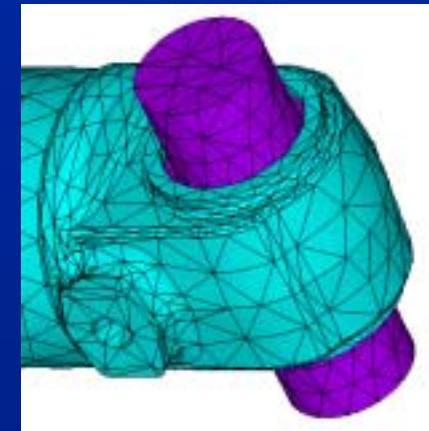
Surface Interaction Models

ANSYS

- Standard contact (unilateral contact)
- Rough contact - no sliding permitted (infinite MU)
- No separation can not open but can slide
- Bonded contact no opening/sliding permitted
 - Assembly contact takes advantage of the bonded contact to glue multiple parts together
- Debonding for modeling crack (Beta)



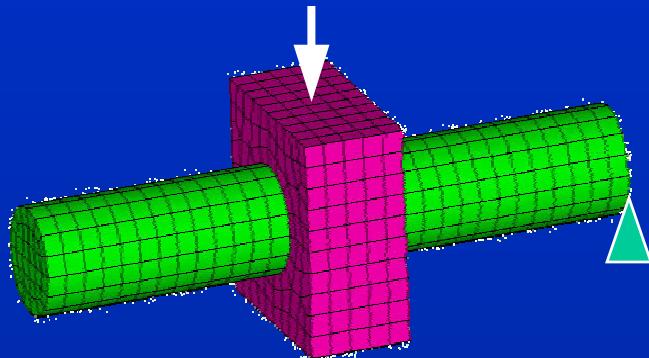
Glue parts A & B
together using
assembly contact



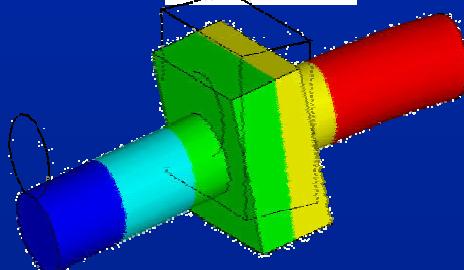
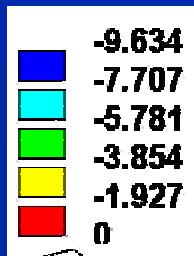
Bonded Contact

ANSYS

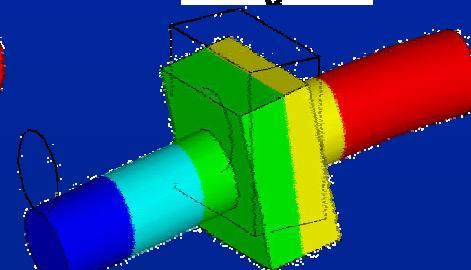
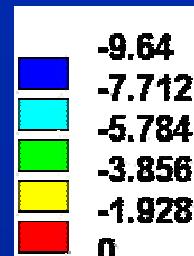
```
/prep7  
et,1,45  
mp,ex,1,2e5  
mp,dens,1,7.8e-9  
cylin,0.5,0.60,0.90  
block,0,10,0,10,25,35  
vovl,all  
esiz,2  
vsweep,all  
vsym,x,all  
vsym,y,all  
numm,node  
numm,kp  
/solu  
ngeo,on  
nsel,s,loc,z,0  
d,all,all  
nsel,s,loc,y,10  
sf,all,pres,200  
alls  
solv
```



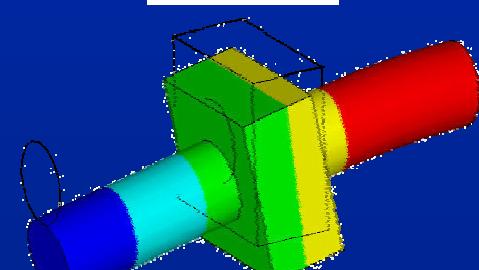
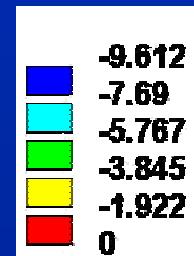
Large deflection



Traditional way



Bonded contact

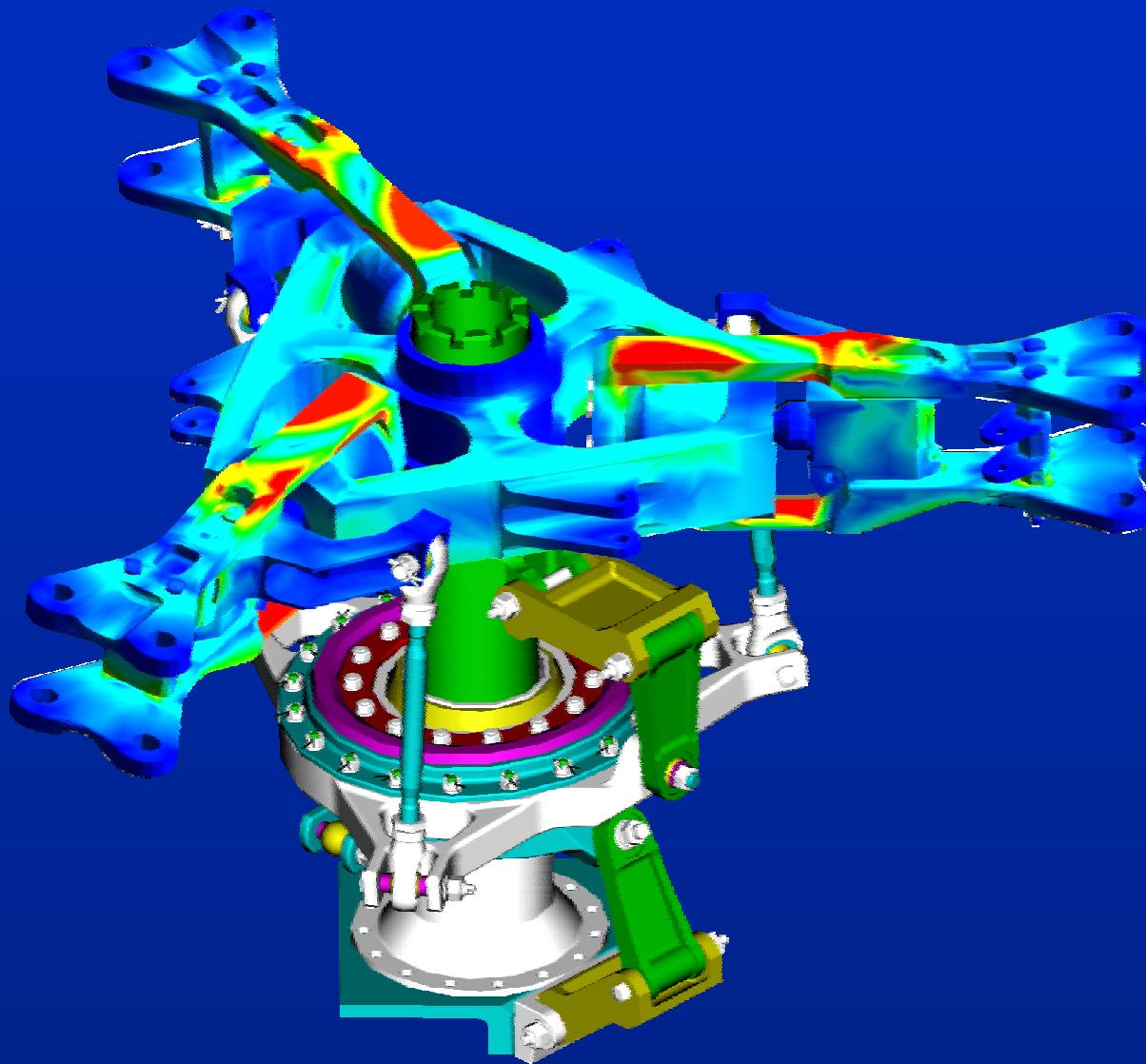


CEINT

CEINT is valid only if the relations between the CE DOFs are linear

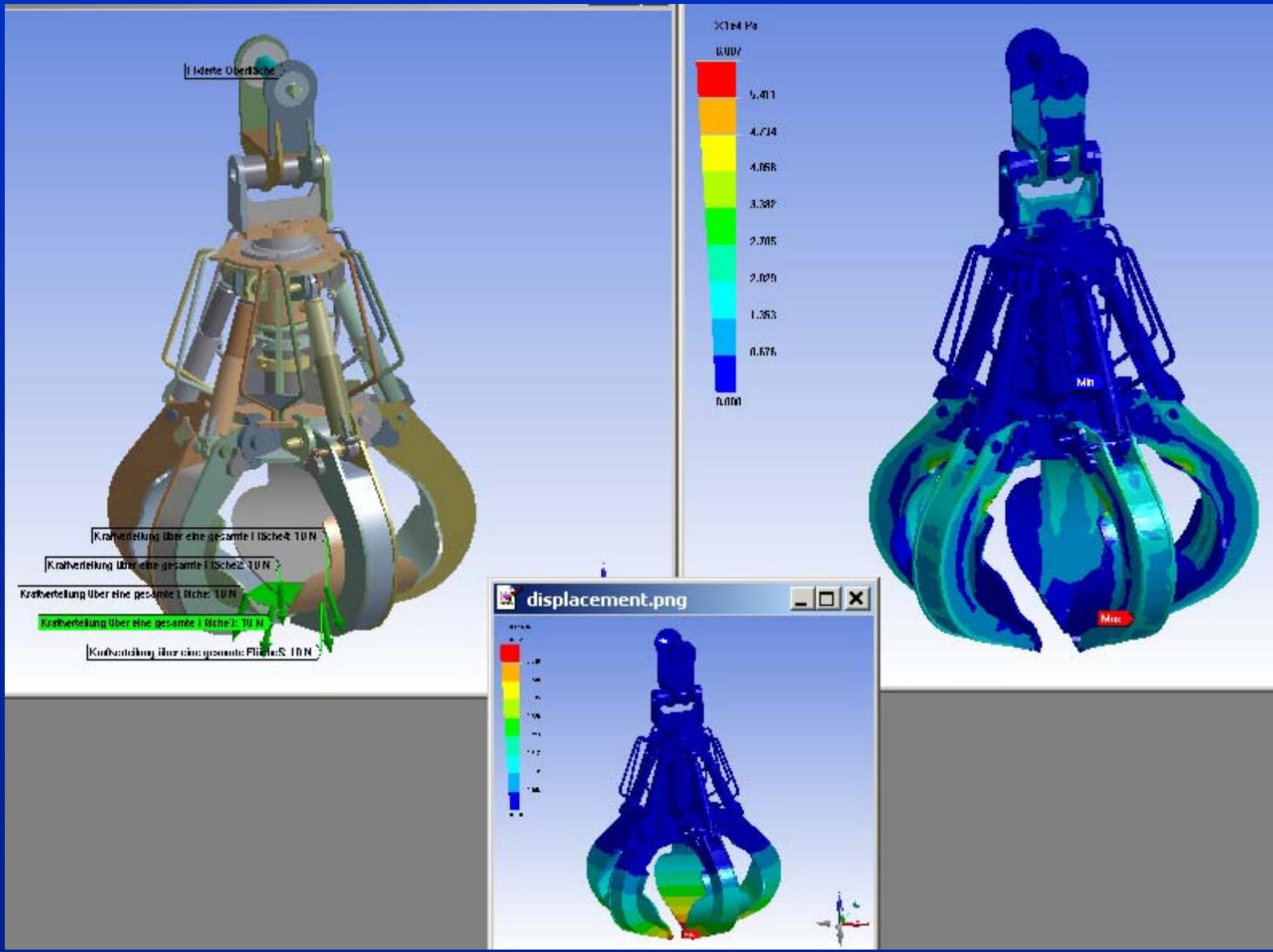
Application: Helicopter Rotor Shaft

ANSYS



Application: Grappling Hook

ANSYS



Bonded contact Based MPC



- A powerful tool to connect incompatible mesh regions
 - Solid-solid, shell-shell, shell-solid assembly
- Multi-point constraints are built automatically
 - Based on surface normal and shape function of target element
 - Updated during each iteration
 - Working properly for larger deformation
 - Solving purely linear contact problems without iteration
 - DOF of the constrained contact nodes is eliminated
- Translational, rotational, temperature, voltage, and magnetic potential can be constrained

3D Assembly Stress Analysis

ANSYS

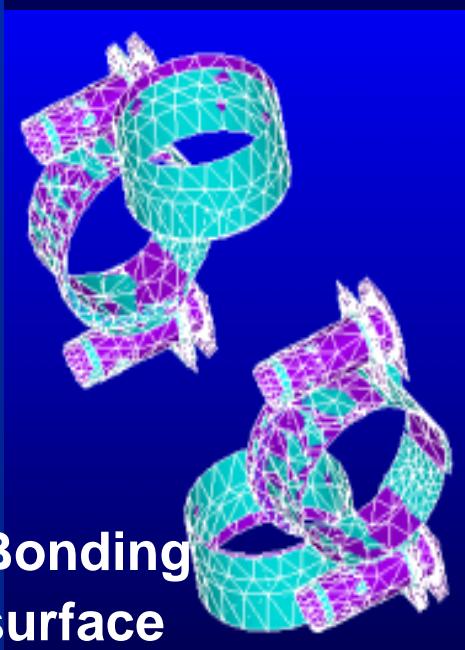
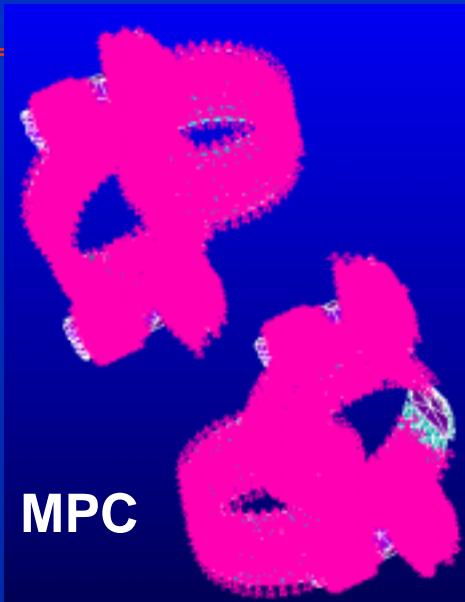
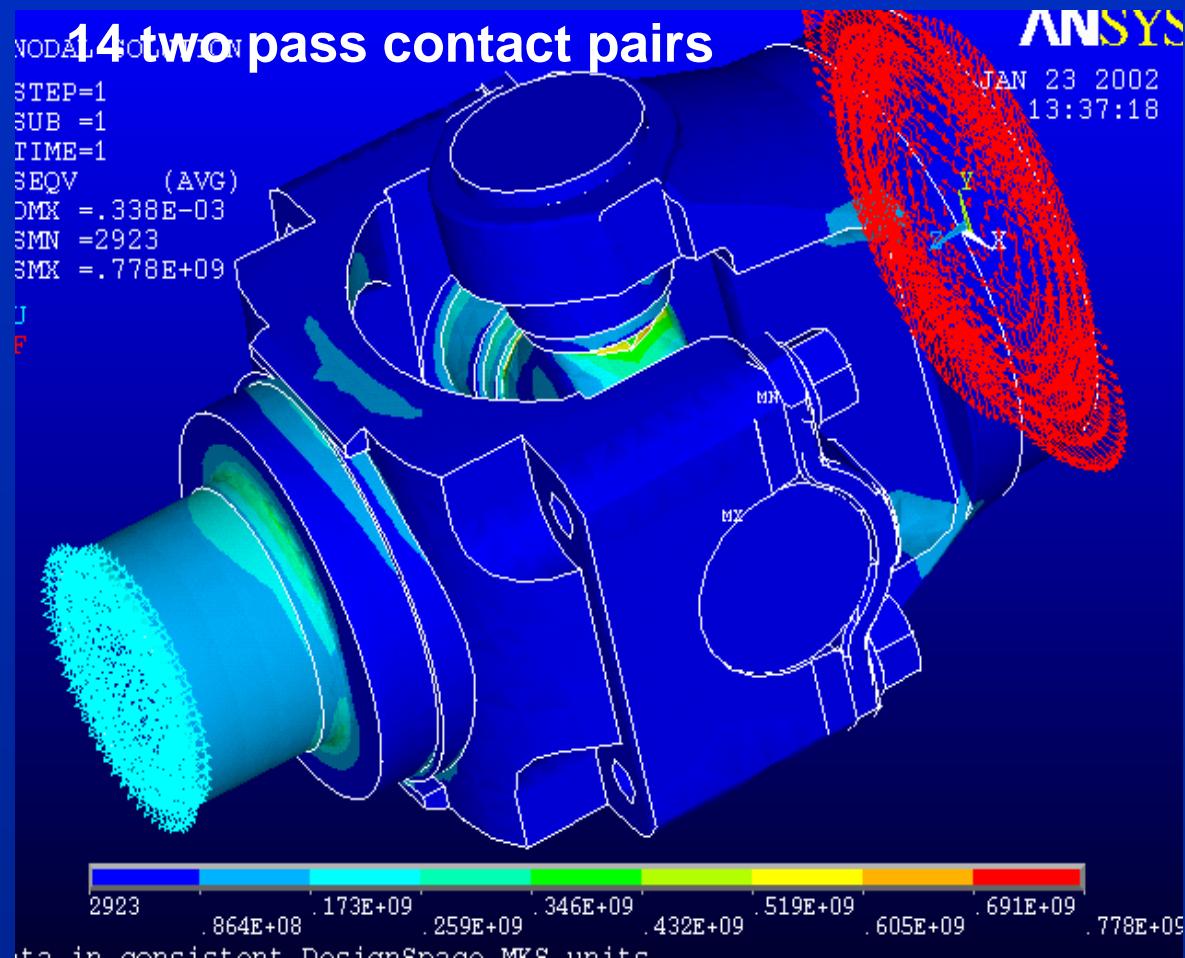
14 two pass contact pairs

NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
DMX = .338E-03
SMN = 2923
SMX = .778E+09

J
F

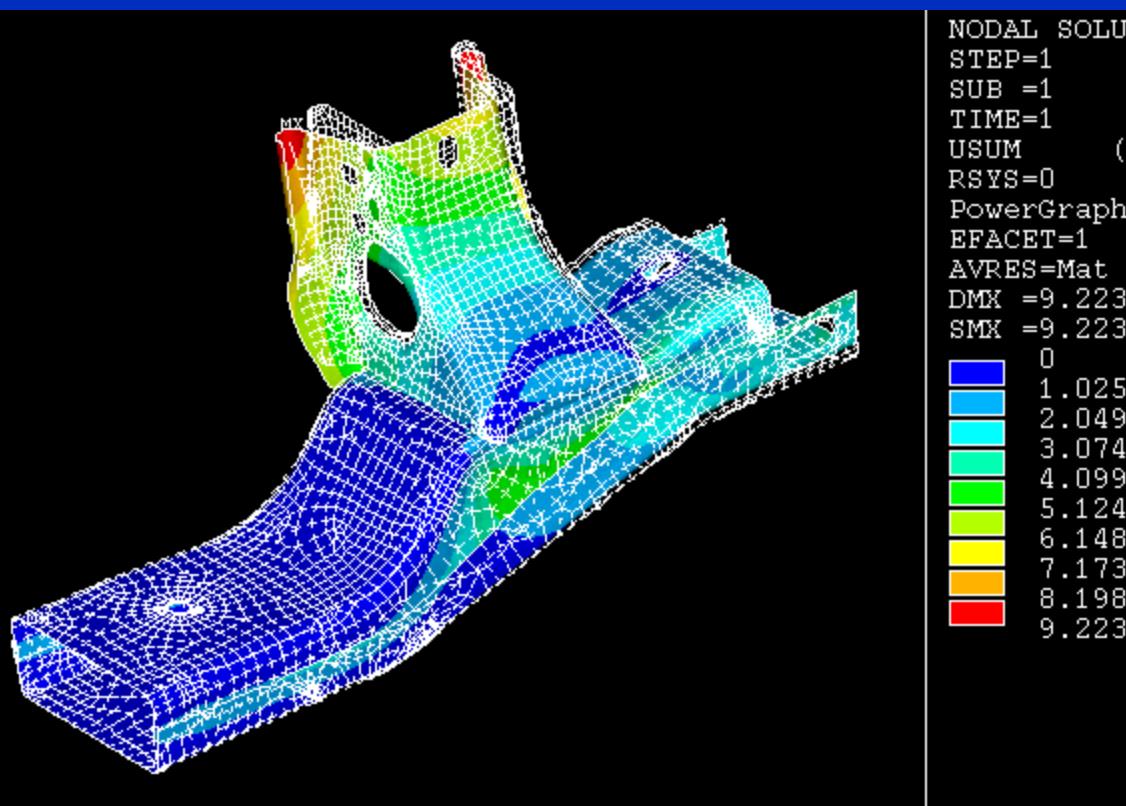
ANSYS

JAN 23 2002
13:37:18

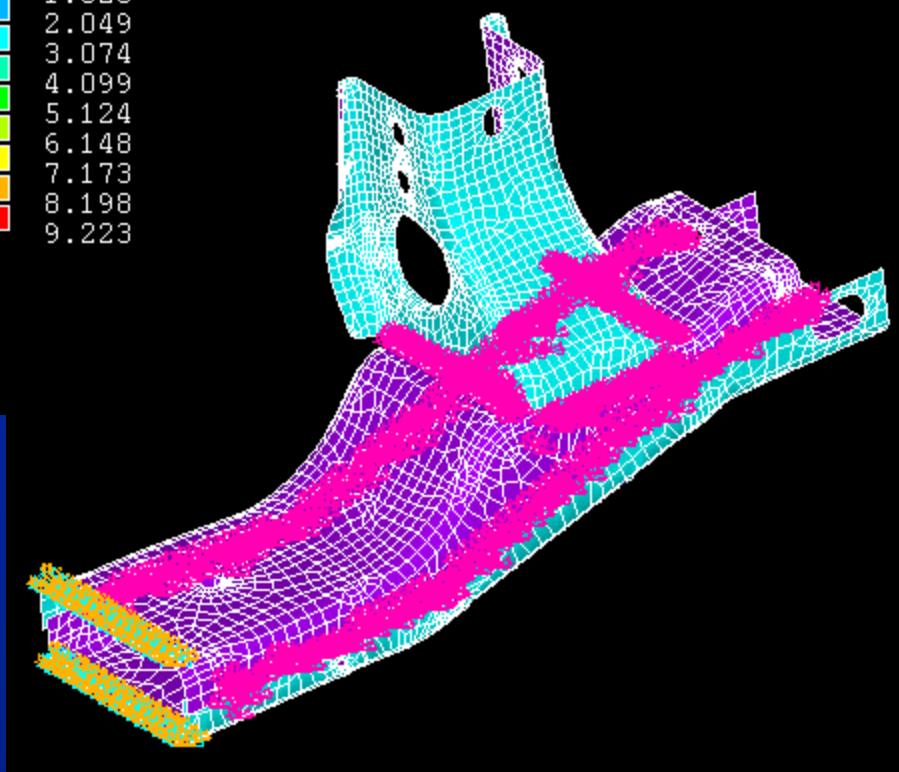


Shell-Shell Assembly

ANSYS

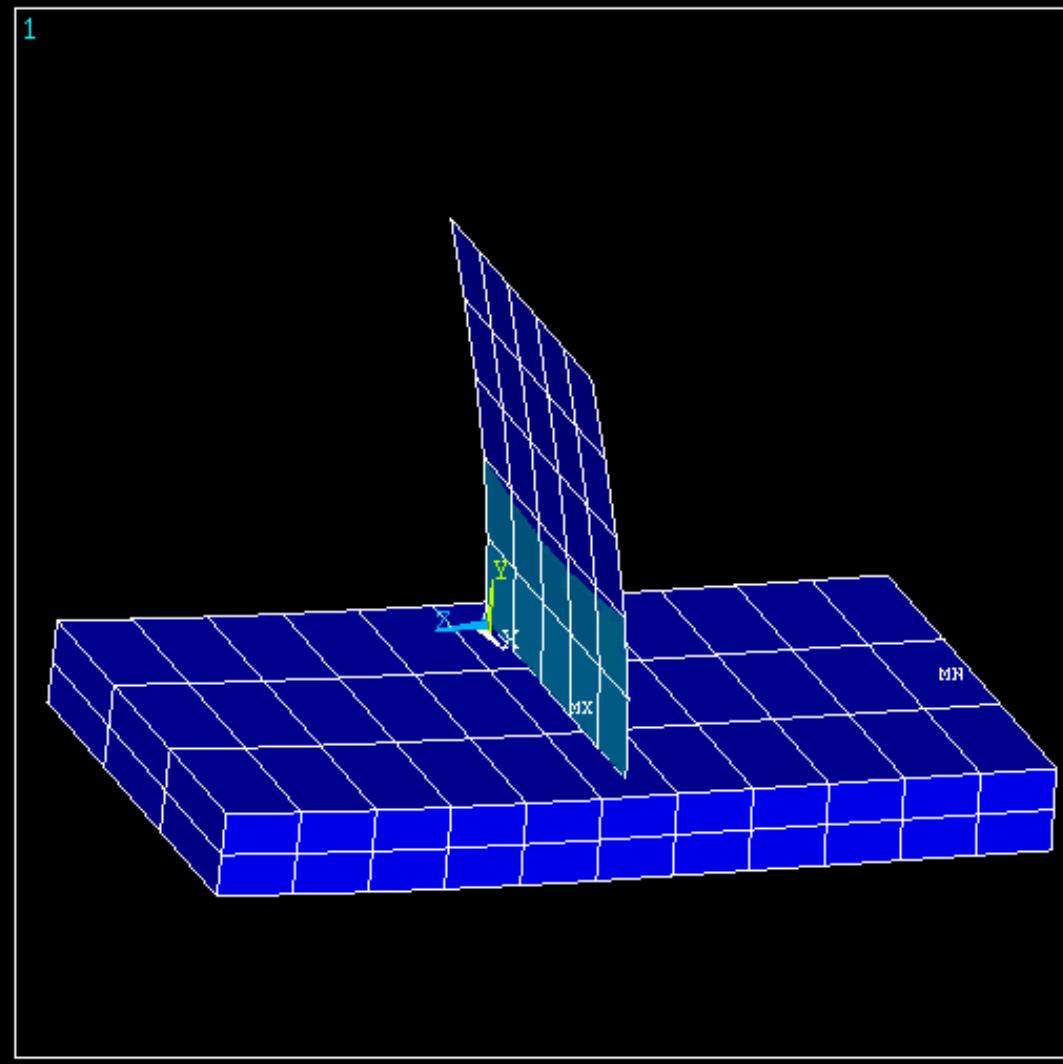


Translational, rotational DOFs
are constrained

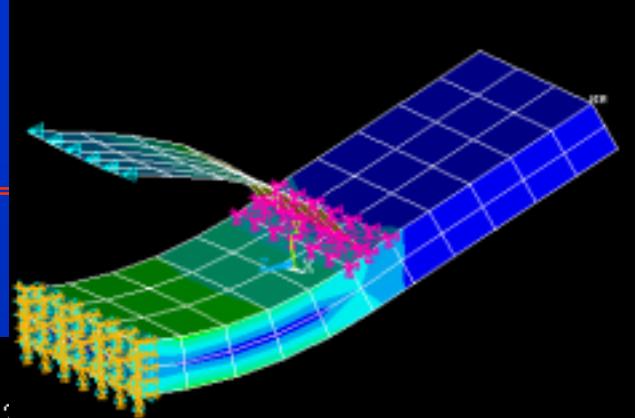


Shell Edge Assembly

14 symmetric contact pairs



ANSYS 6.1
MAR 5 2001
16:42:33
NODAL SOLUTION
STEP=1
SUB =1
TIME=.2
SEQV (AVG)
PowerGraphics
EFACET=1
AVRES=Mat
DMX =.20004
SMN =6.089
SMX =19607
6.089
13177
26348
39519
52691
65862
79033
92204
105375
118546



Multi-Physical Contact Element

CONTA171-175

A Cutting Edge Technology

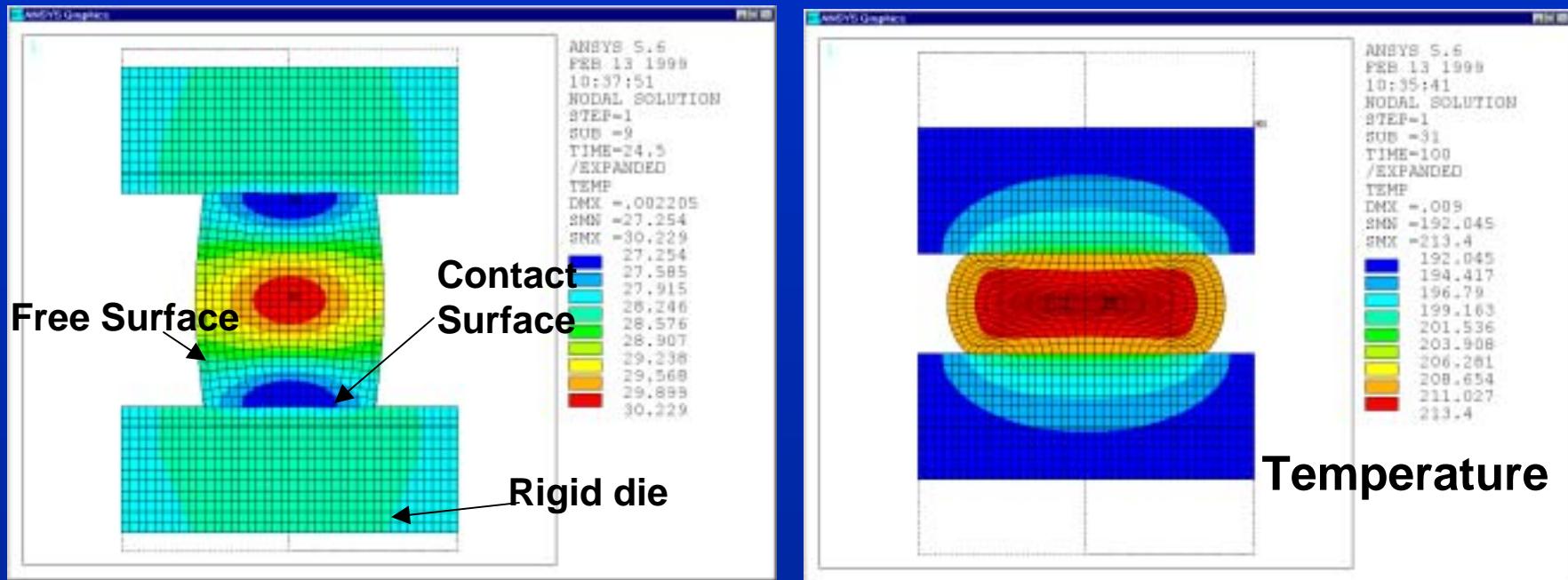
Multi-Physics Contact



- Heat/electric conduction between contacting surfaces
- Heat convection and/or radiation
- Heat generation due to
 - Frictional dissipated energy
 - Electric current
- Electric charge across the contacting interface
 - Piezo-electric analysis
 - Electrostatic analysis
- Magnetic flux across the contacting interface

Application: Upsetting of a Billet

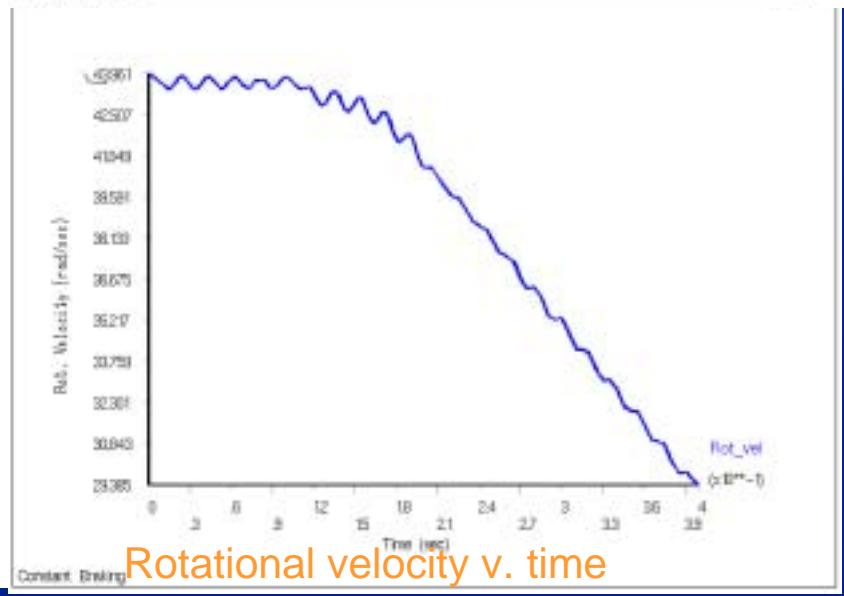
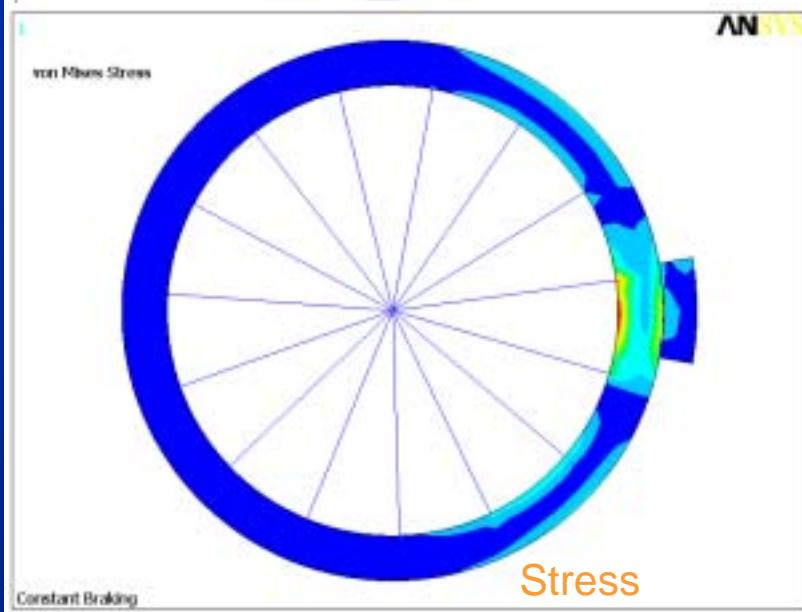
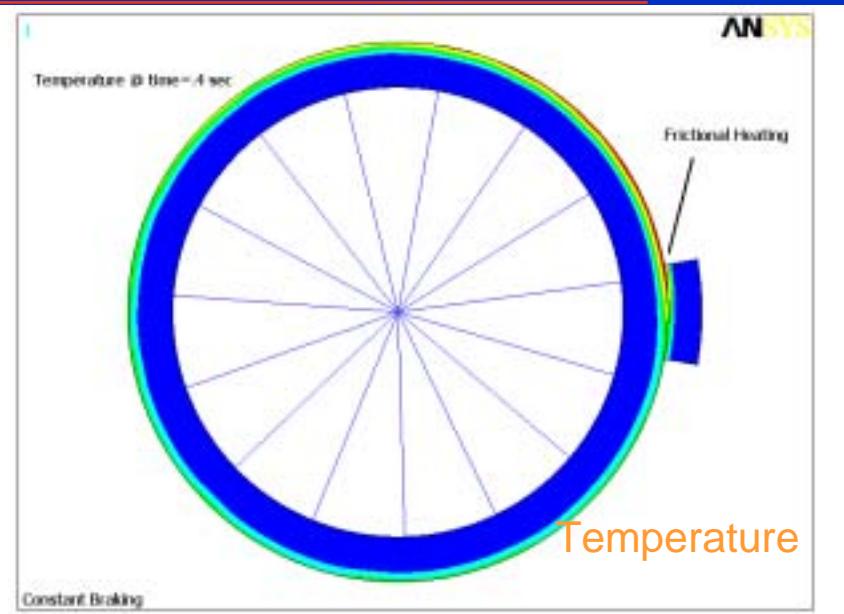
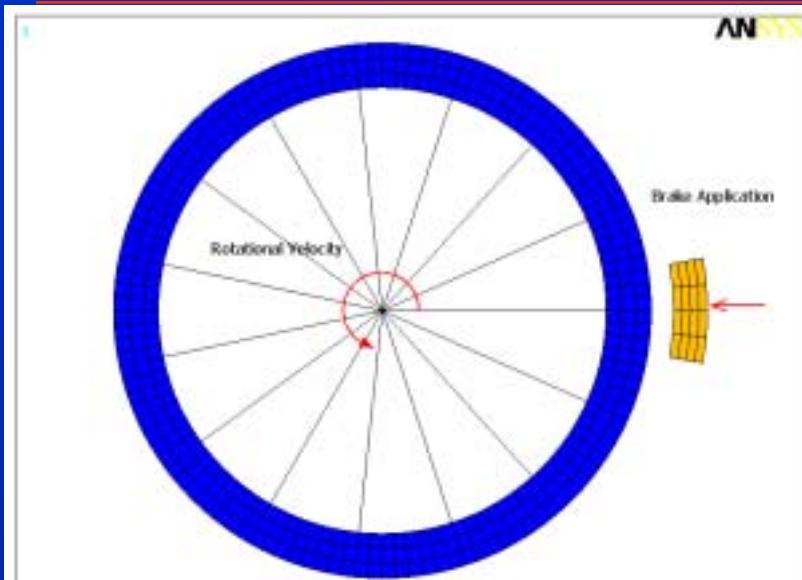
ANSYS



- Heat generation due to plastic work
- Heat convection and radiation on free surface
- Heat conduction and frictional heating on contact surface

Application: Wheel Brake

ANSYS



Heat Sink Analysis with Radiation

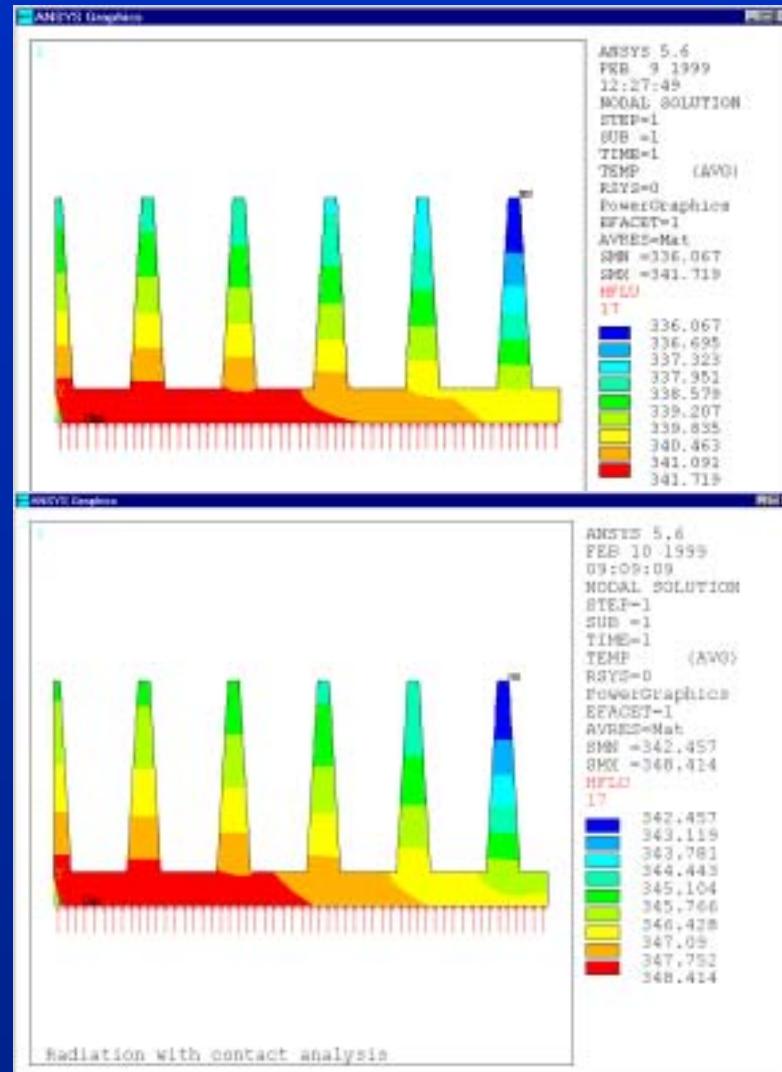
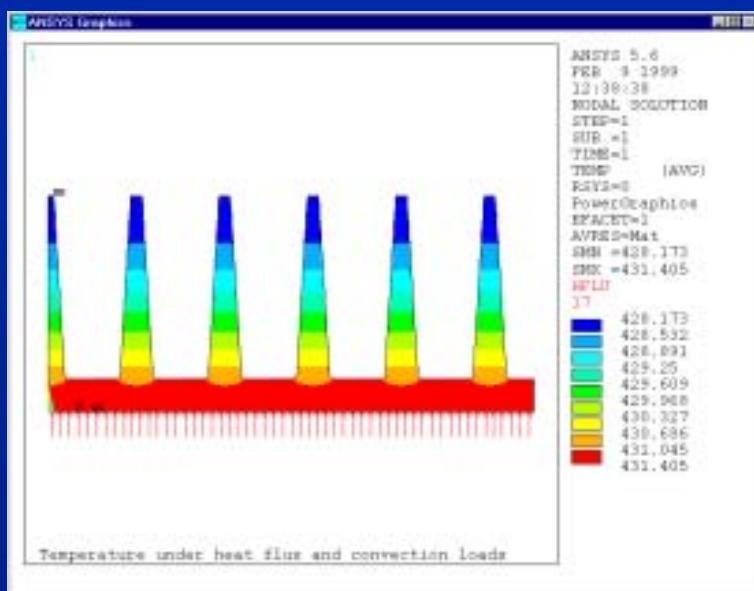
Loads:

Heat flux

Convection

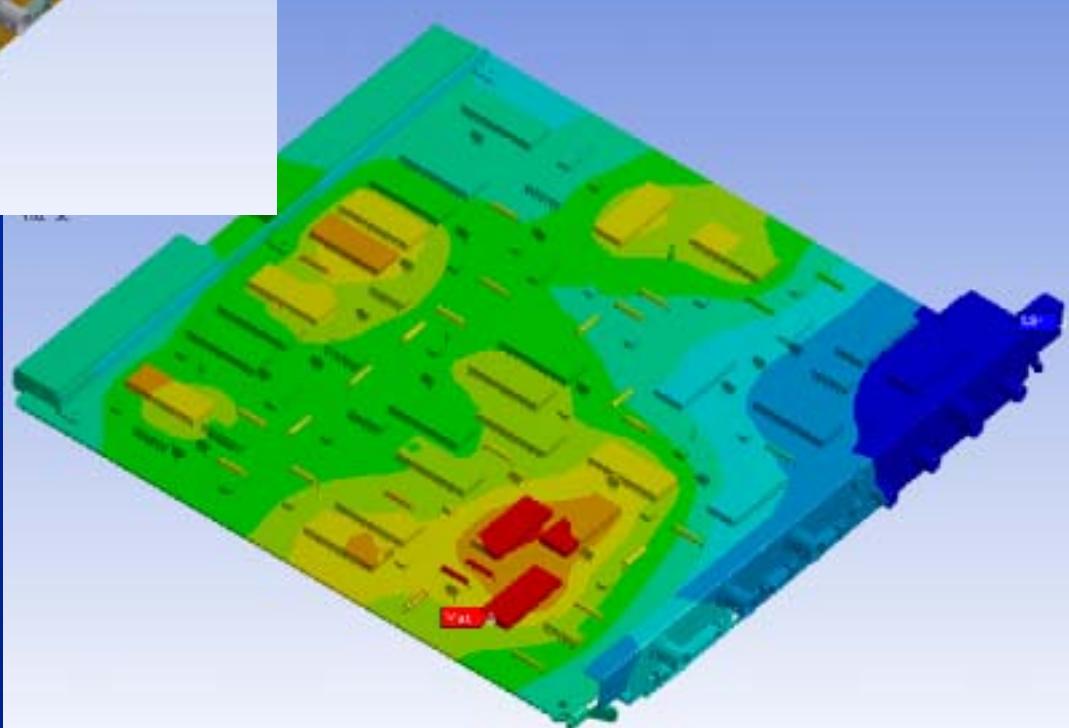
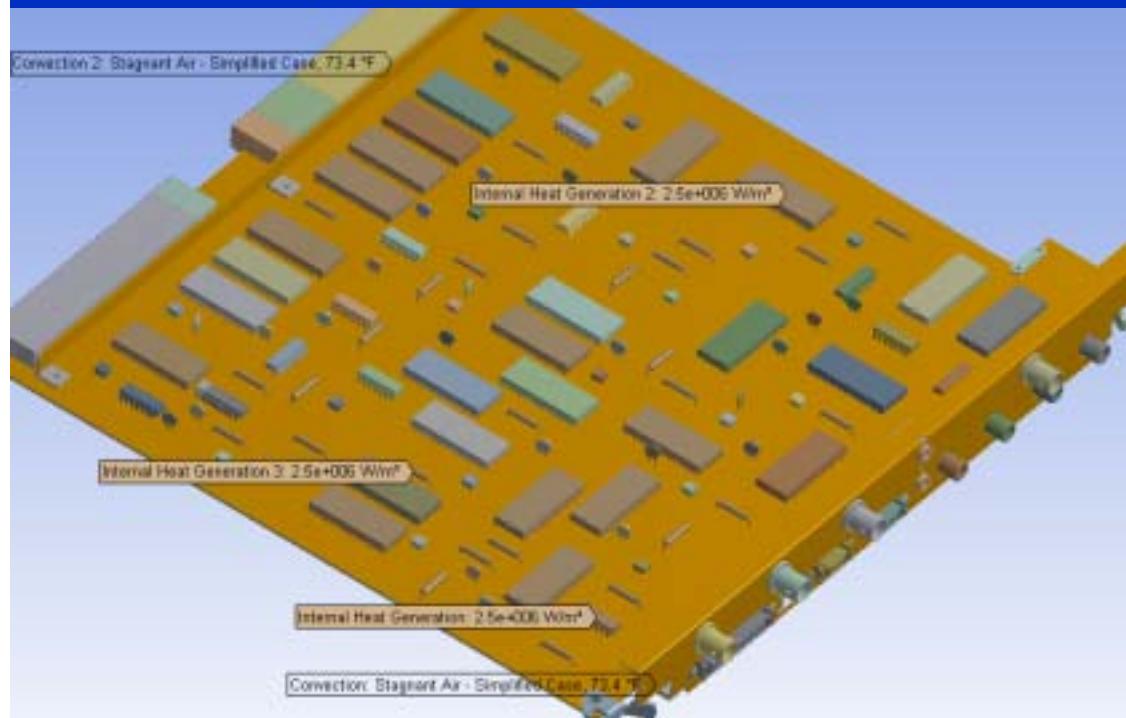
Radiation

Surrounding Temperature = 90



Application: Circuit Board

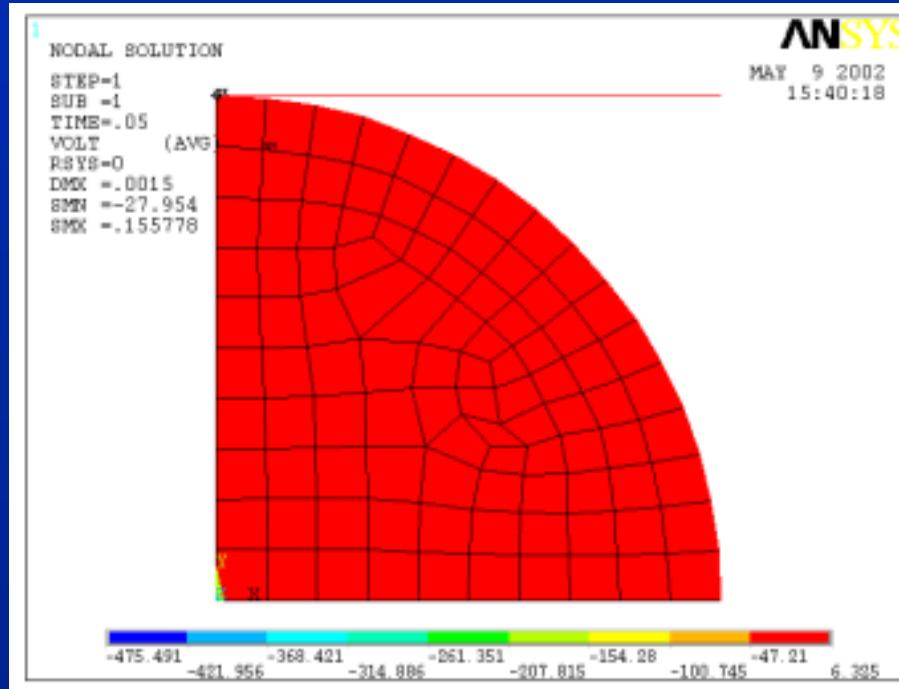
ANSYS



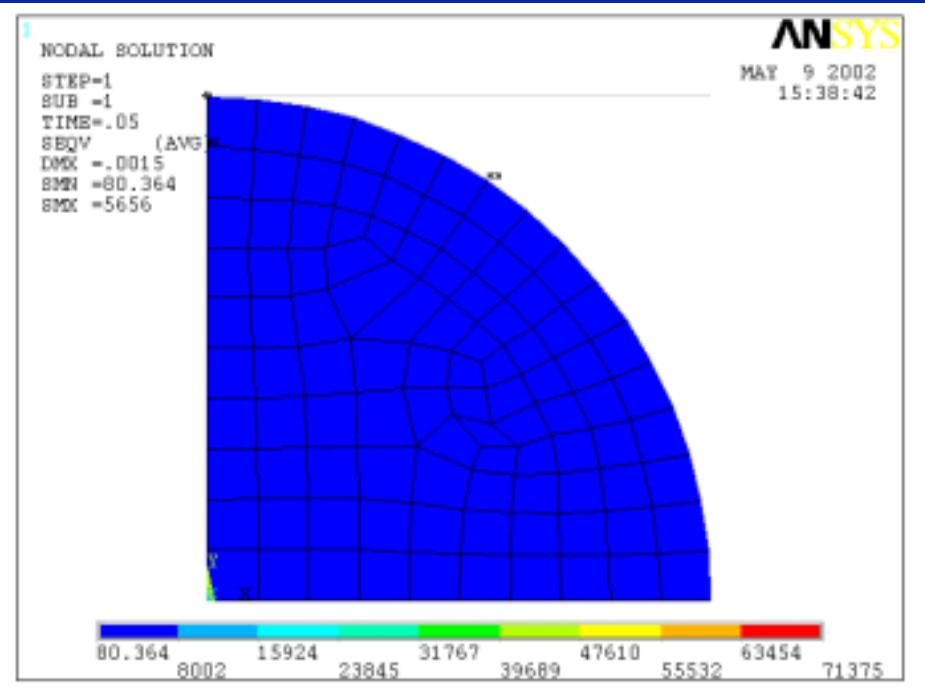
Piezo-electric Environment

ANSYS

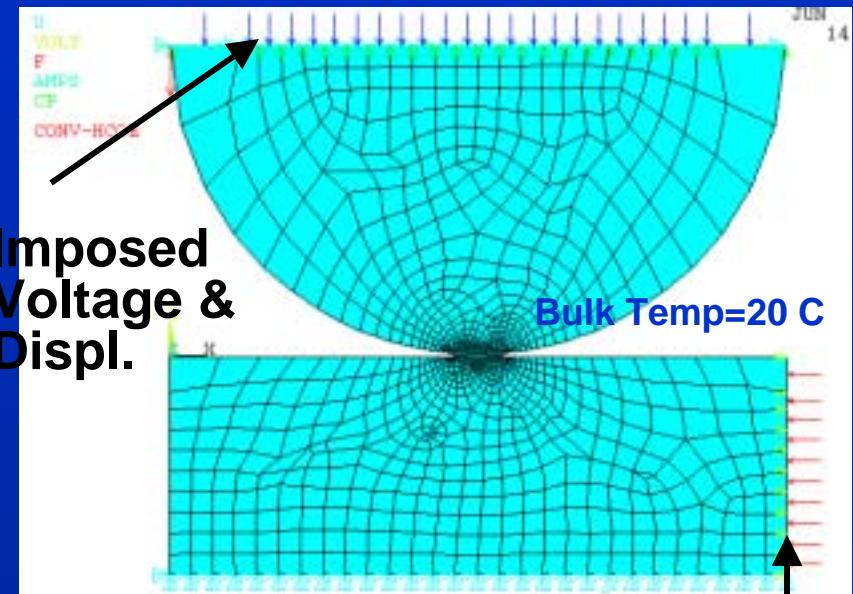
VOLT DOF



VM Stress



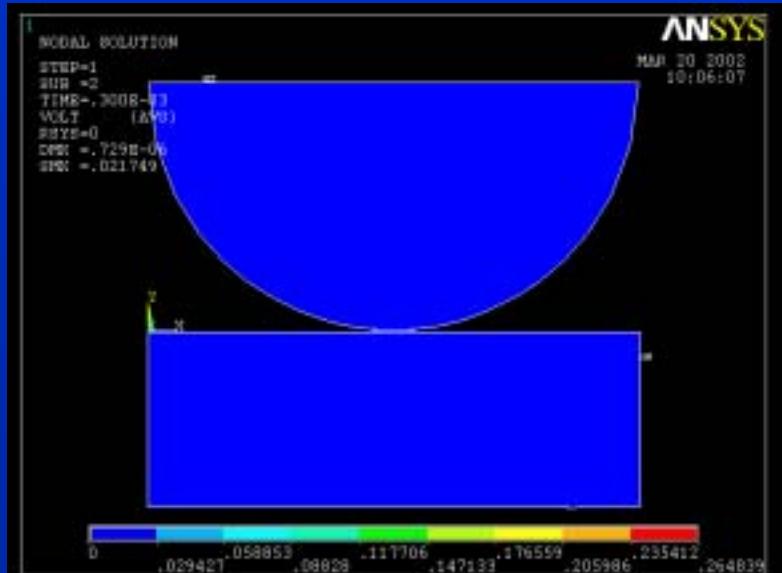
Application: Electric Welding



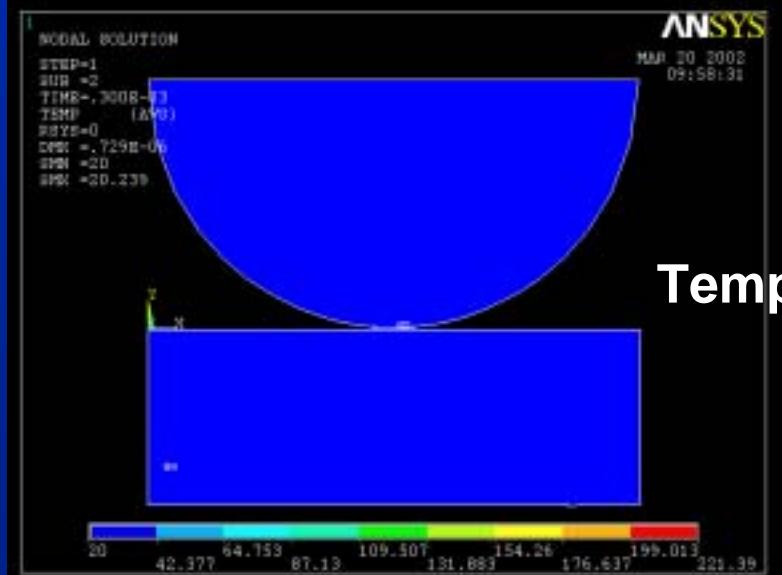
Imposed
Voltage &
Displ.

Grounded

Transient analysis (Time: 0-0.1s)



Voltage

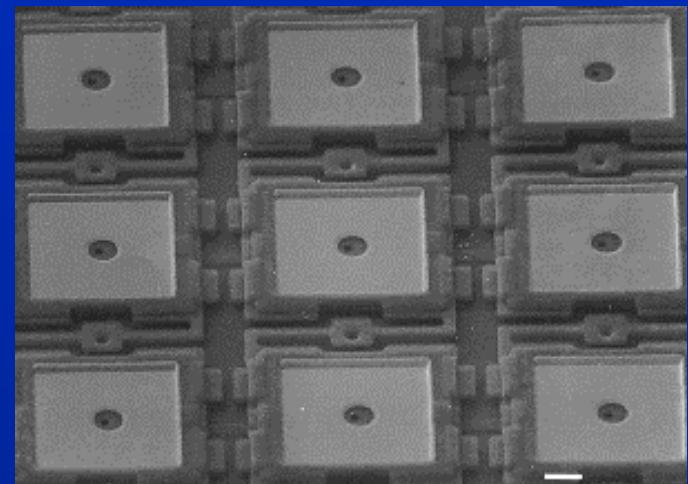
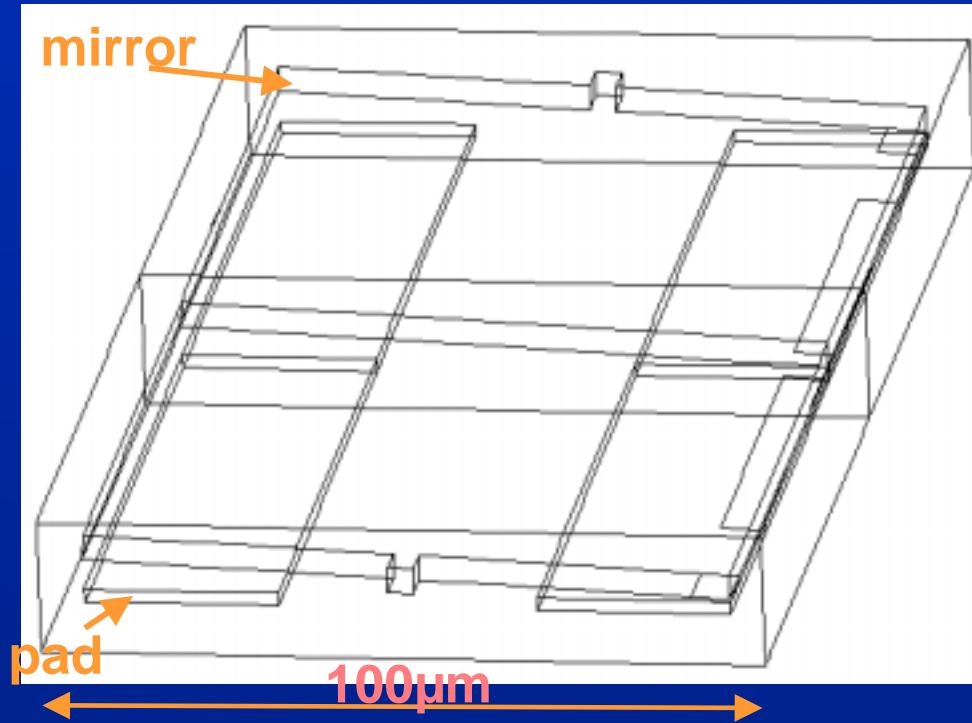


Temperature

Application: Electrostatic Micro Mirror

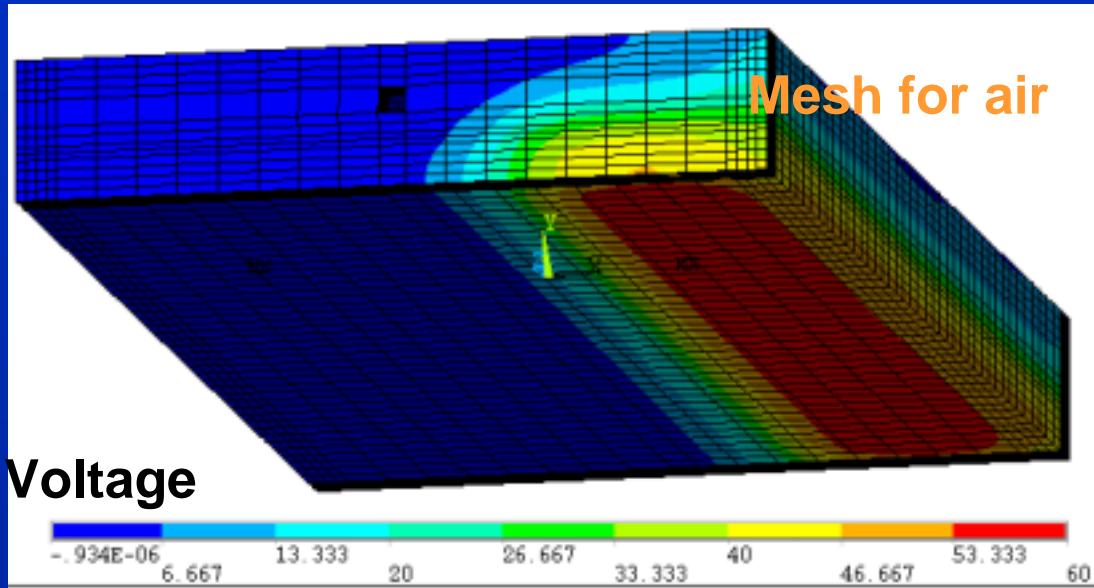
ANSYS

- Using sequential electrostatic-structural coupling procedure
- Voltage difference between mirror and pad creates unbalanced electrostatic forces

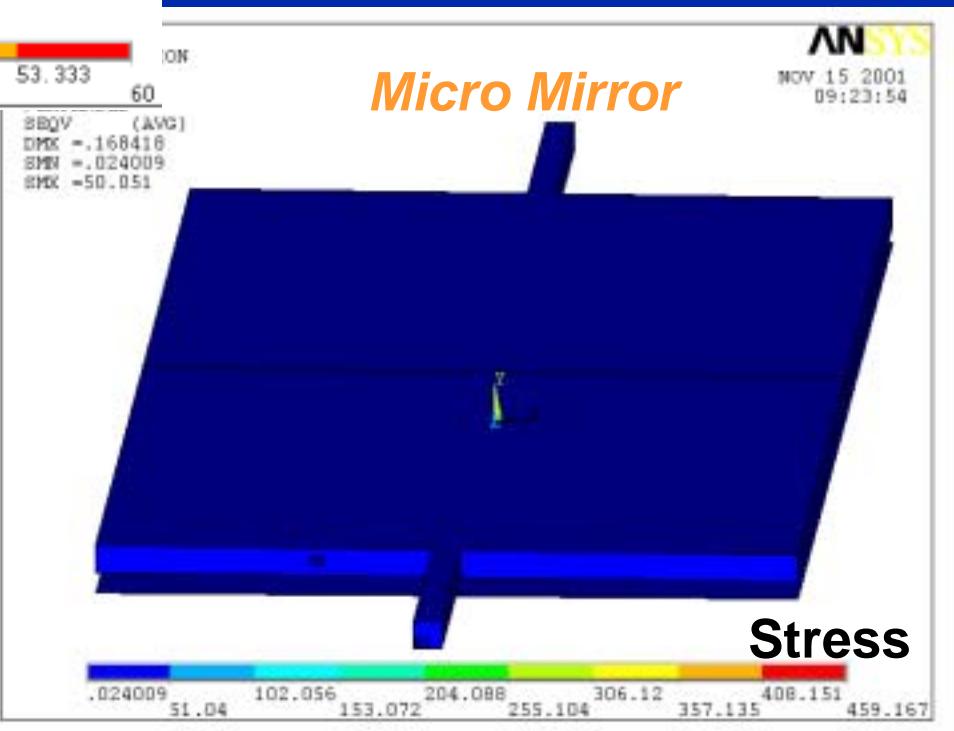


Application: Electrostatic Micro Mirror

ANSYS

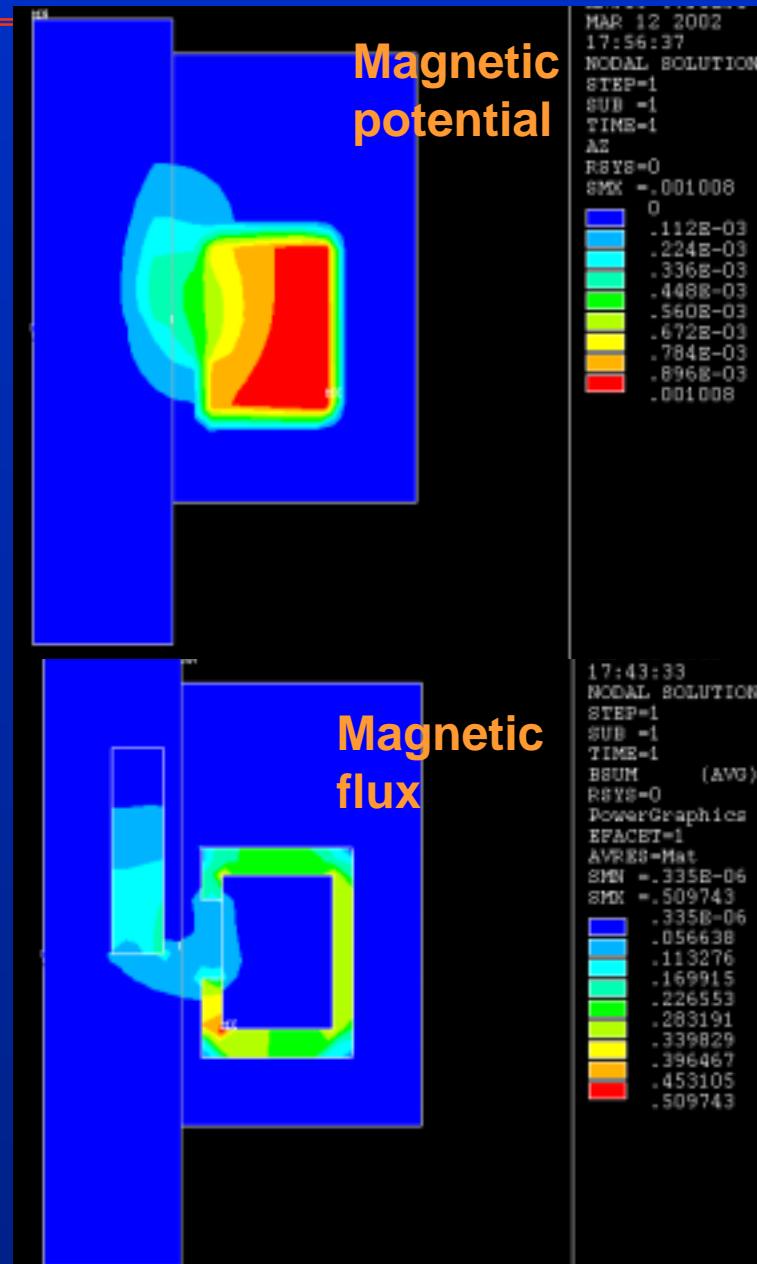
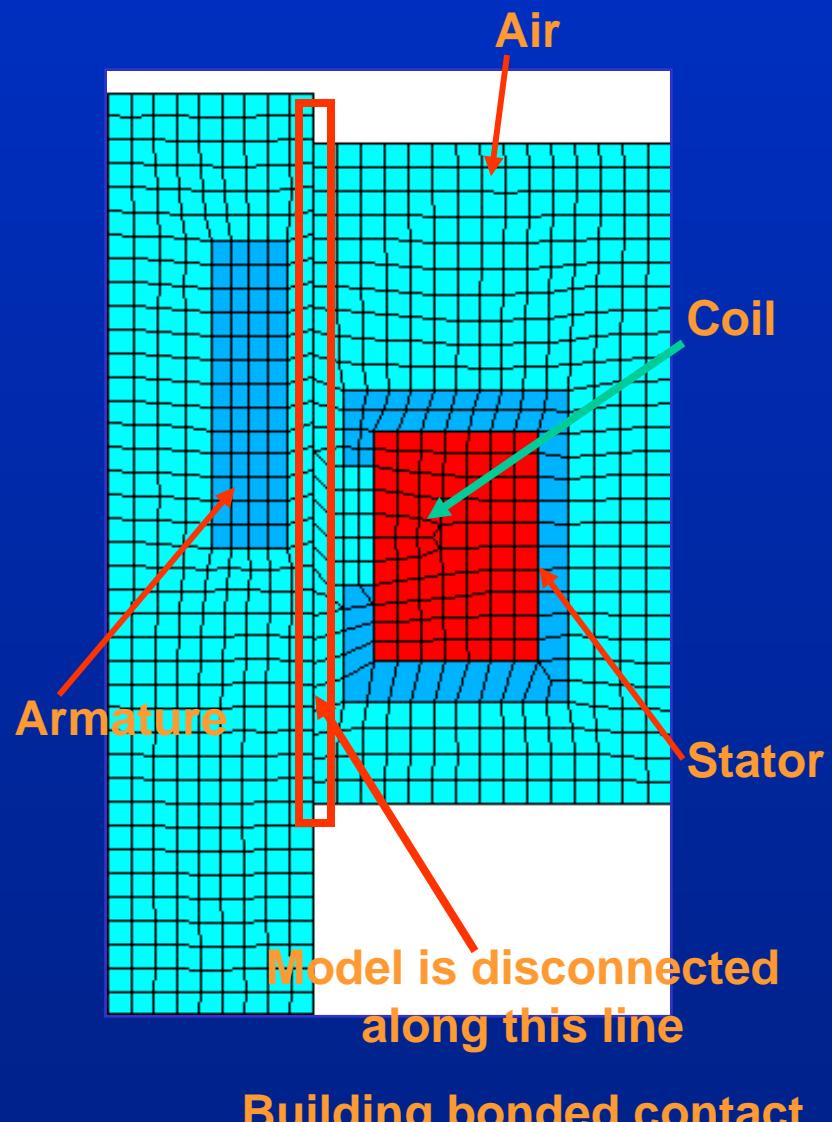


- Simulation with iterative process
 - Electrostatic solution to get force
 - Structural solution with contact
 - Morphing of electrostatic mesh



Axisymmetric DC Actuator

ANSYS



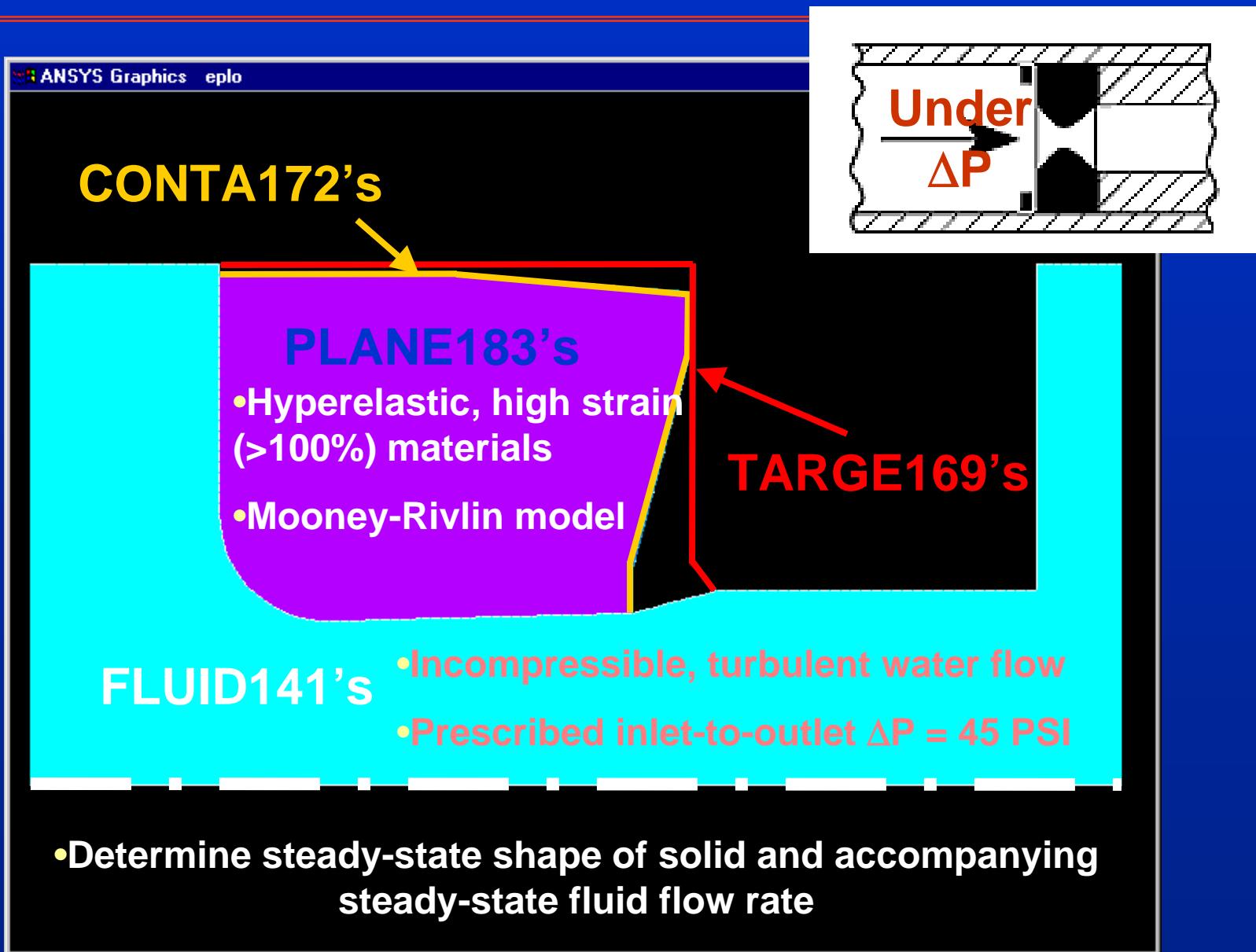
Fluid-Structure Interaction



- **Fully-automated multiphysics solution for**
 - Fluid-structure interaction, fluid-thermal-electric interaction
 - Full support for all nonlinearities: Geometric, material, contact
 - Fully-implicit time-stepping scheme:
 - FLOTTRAN Element Birth and Death:
 - Suitable for FSI problems involving contact between immersed, moving structures
 - Fluid elements may be automatically deactivated as surfaces come into contact (e.g., valve closes), or reactivated as they separate (e.g., valve opens)
- **Extremely wide set of applications**

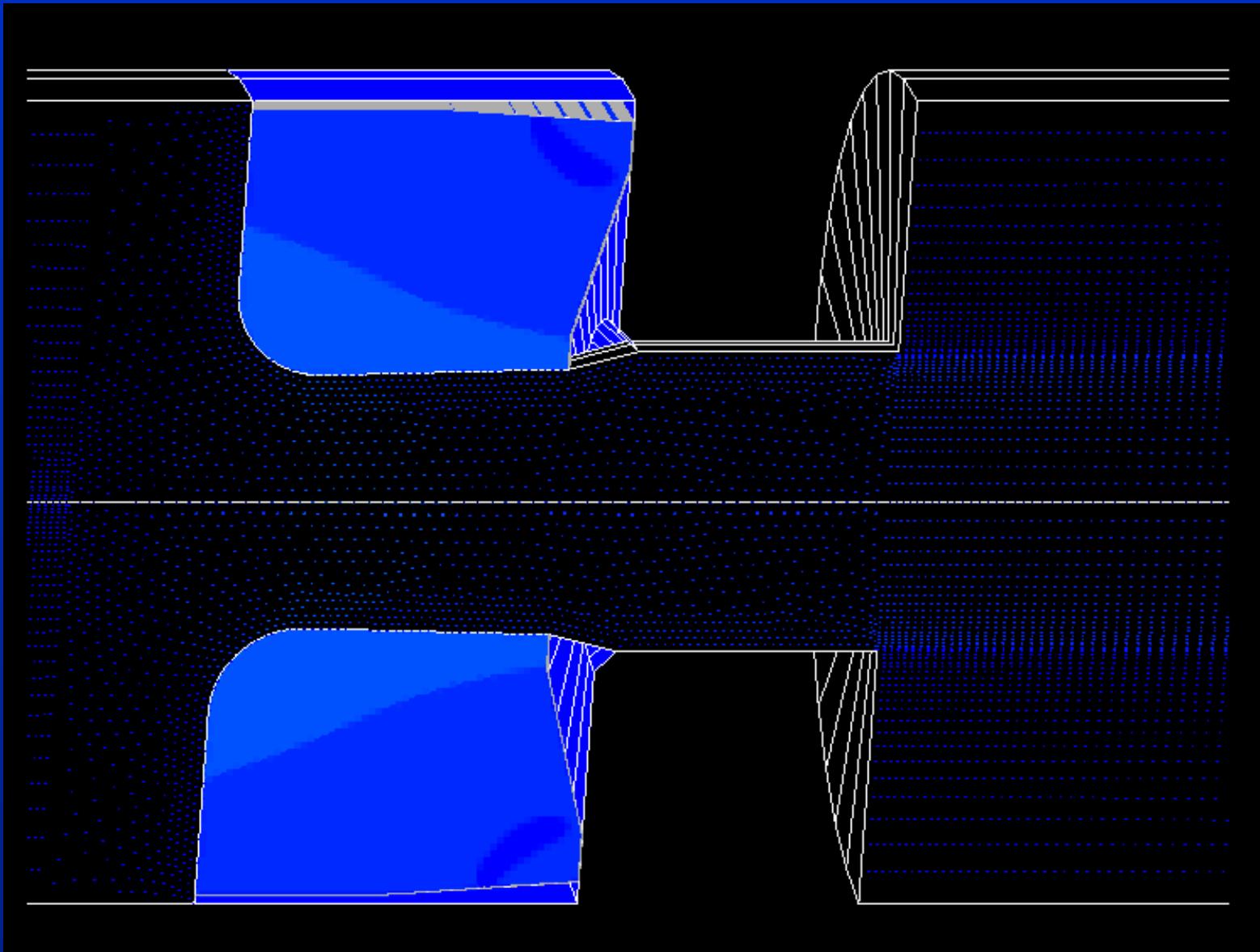
Deformable Flow Control Device

ANSYS



Deformable Flow Control Device

ANSYS



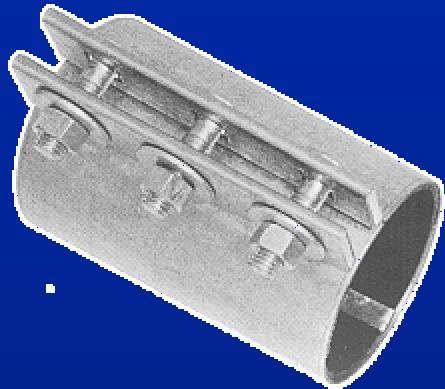
Bolt Pretension Element PRETS179

A State of Art

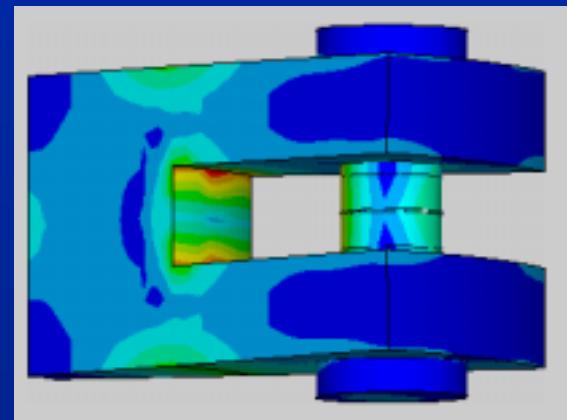
Bolt Pretension

ANSYS

- Whenever you model a bolted structure, it might be important to include the **pretension** (or preload) in the bolt caused by the tightening of the bolt.
- ANSYS provides a convenient way to simulate bolt pretension:
 - Pretension elements PRETS179
 - Automatic pretension mesh generation
 - Load management for pretension sequence of multiple bolts



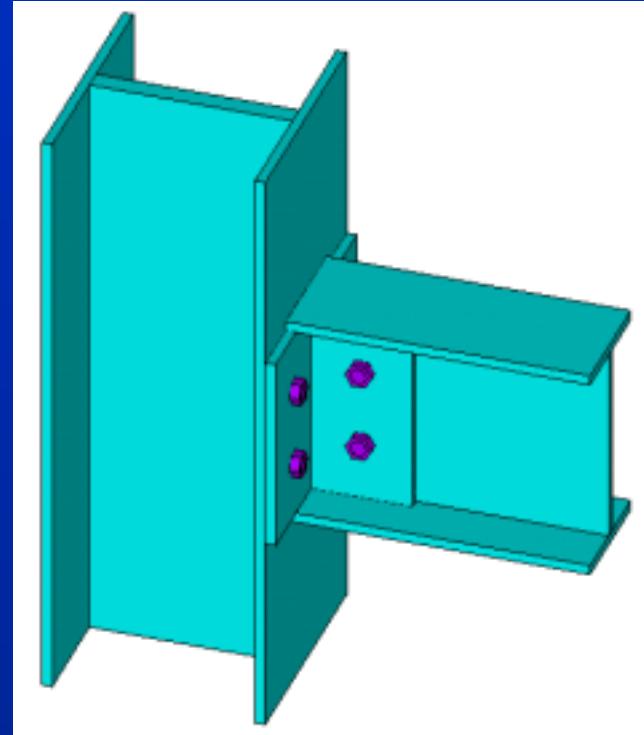
Stresses due
to specified
pretension in
bolt



Bolt Pretension: Traditional Ways

ANSYS

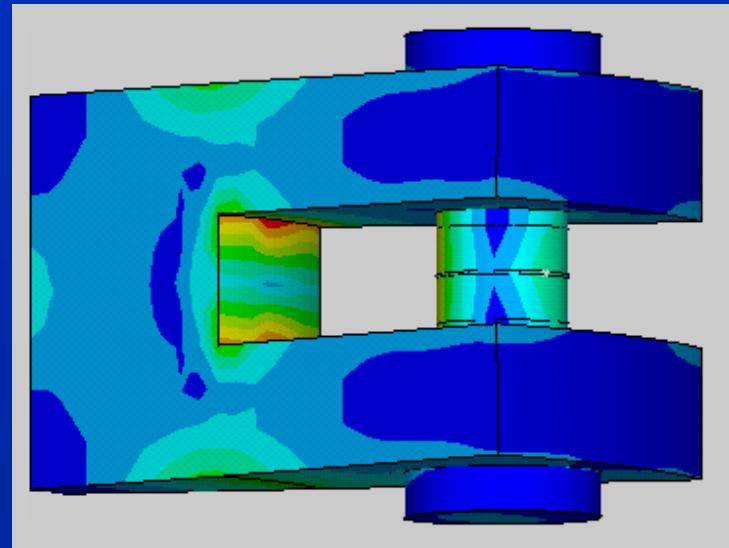
- There was no automated way to prescribe the pre-load. It was modeled through
 - Contact initial interference
 - Thermal expansion
- Manually iteration process is required
- It is difficult to control and monitor the variation of the bolt force during whole modeling



Bolt Pretension : Overview

ANSYS

- An automated way to specify bolt pretension
- Useful for creating, managing, and loading structures having multiple pretensioned bolts (no limit on the number of bolts)
- Replaces previous trial-and-error techniques
- Automated generation capability

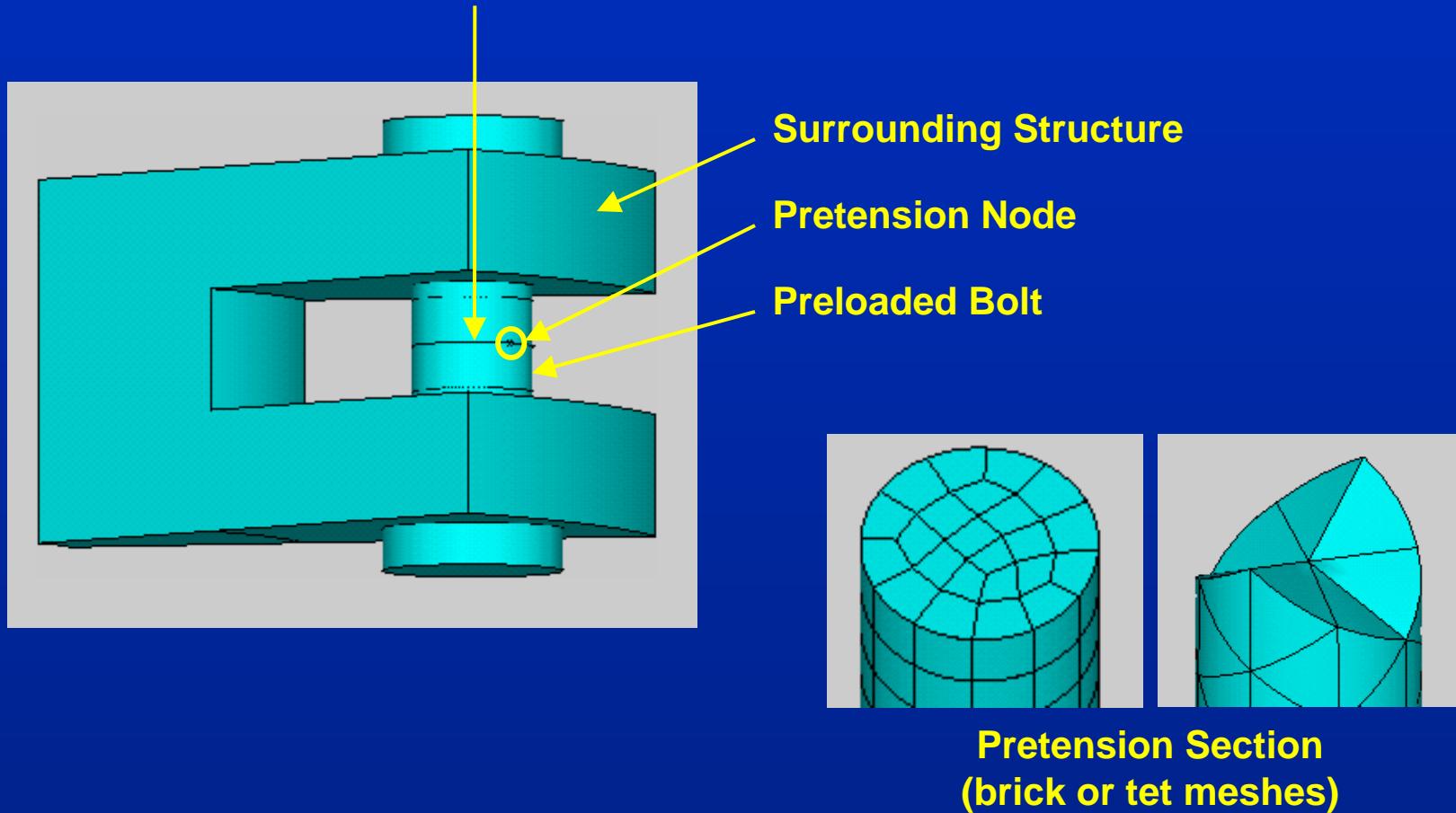


Stresses Due to Specified Pretension in Bolt

PRETS179 Element

ANSYS

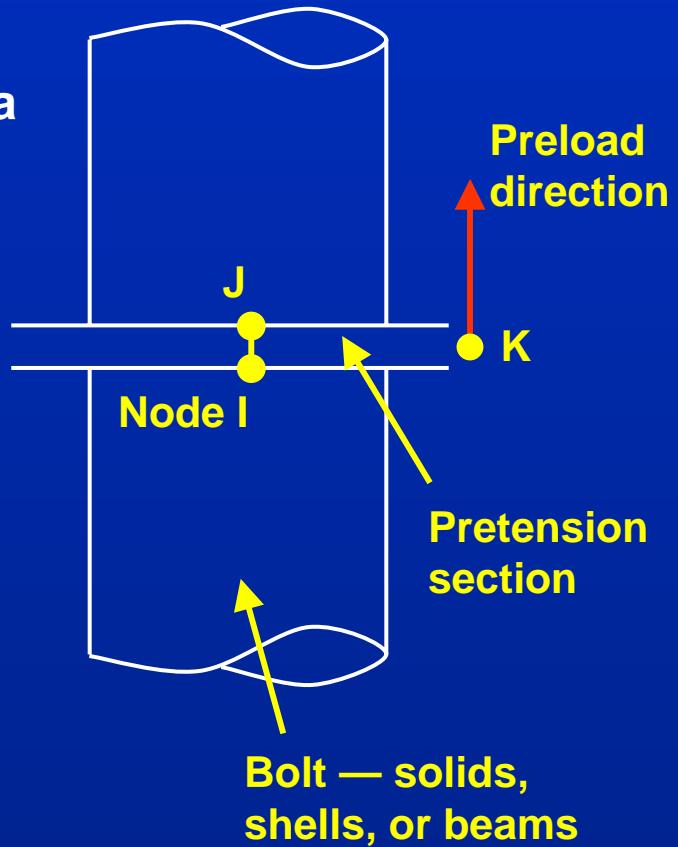
- Pretension elements (PRETS179) apply the specified preload across a *pretension section*.



PRETS179 Element

ANSYS

- Features of the pretension element:
 - A set of pretension elements is identified as a “section”.
 - 2-D or 3-D line element that acts like a “hook” connecting two halves of a bolt.
 - Nodes I, J are the end nodes, usually coincident.
 - Node K is the *pretension node*:
 - Location is arbitrary.
 - Has one DOF: UX.
 - Used to define the preload, as an FX force or UX displacement.
 - Actual line of action is in pretension load direction



- **Features of the pretension element (continued):**
 - Preload direction is constant – it does not update for rotations. It can be re-defined during load steps.
 - No material properties or key options
 - Underlying bolt elements may be solids, shells, or beams, lower or higher order.
 - The DOF: translations, rotations, temperature, voltage which are detected internally based the DOF of underlying elements.
 - Elements created automatically using GUI-based procedure.

Typical Procedure



- Five main steps:
 1. Import or create the geometry, including the bolt(s) and the surrounding structure.
 2. Mesh all parts.
 3. Create the pretension elements.
 4. Apply loads and solve using multiple load steps:
 - Load step 1 for the bolt preload
 - Load step 2 to “fix” the bolt length
 - Load step 3 for other loads on the structure
 5. Review results.
- We will expand on steps 3 and 4 next.

Typical Procedure



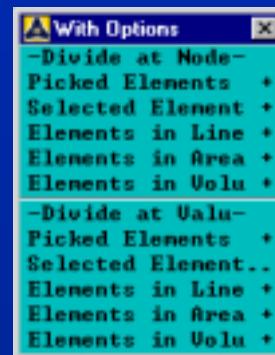
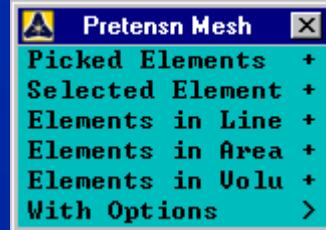
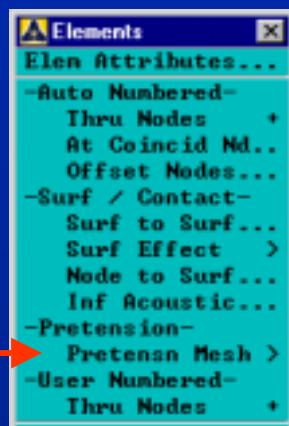
Creating the Pretension Elements

- After all parts of the structure, including the bolt(s), have been meshed, the next step is to create the pretension elements.
- Two options:
 - Using **PSMESH**
 - Fastener must be meshed as one piece.
 - PSMESH will cut the fastener in two and generate the pretension section together with the pretension elements.
 - Elements at coincident nodes, EINTF (not discussed here)
 - The fastener must be meshed in two separate pieces.
 - Requires a matching node pattern.

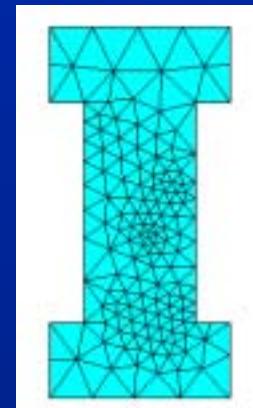
Typical Procedure

ANSYS

- The menus provide a wide variety of methods to create the pretension mesh.
 - Preprocessor > Create > Elements > - Pretension –
Pretensn Mesh >
- We will illustrate the procedure using the With Options > pick.



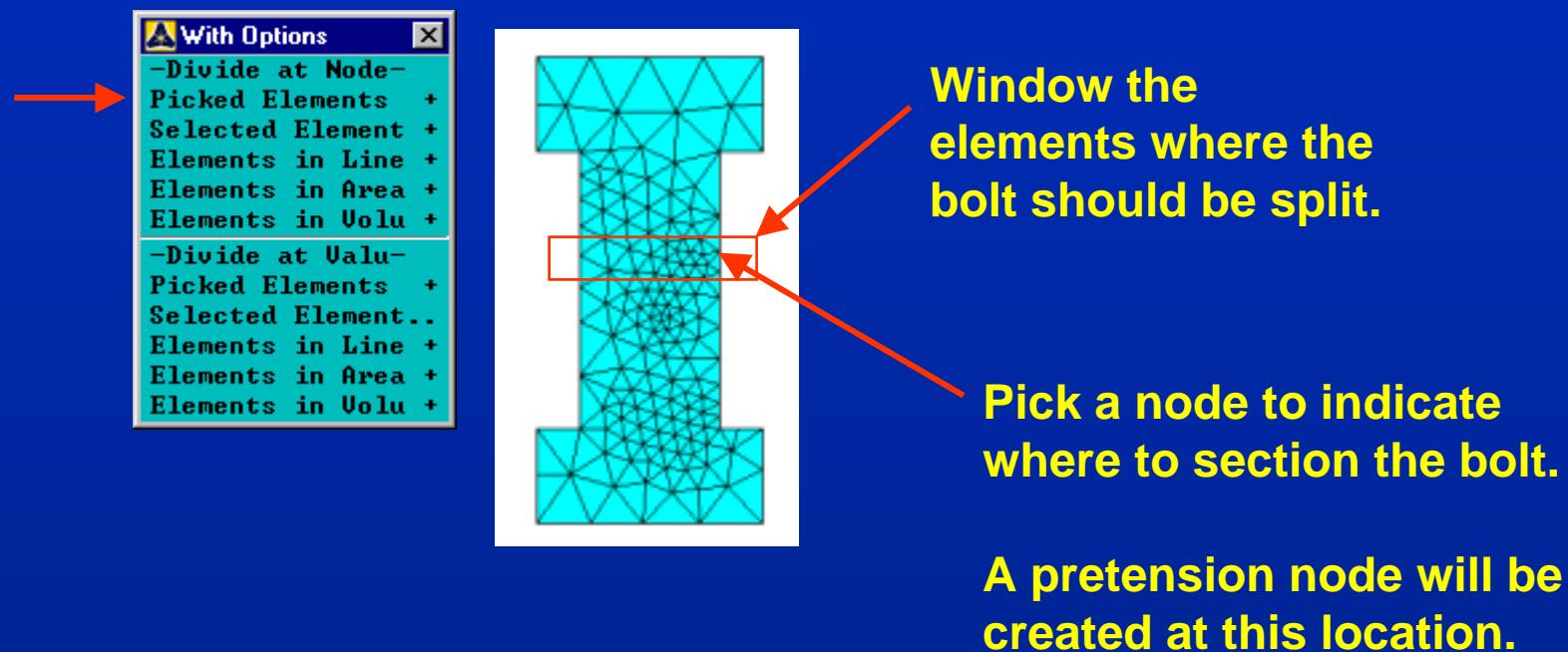
ANSYS splits the bolt shown
and inserts the necessary
pretension elements.



Typical Procedure

ANSYS

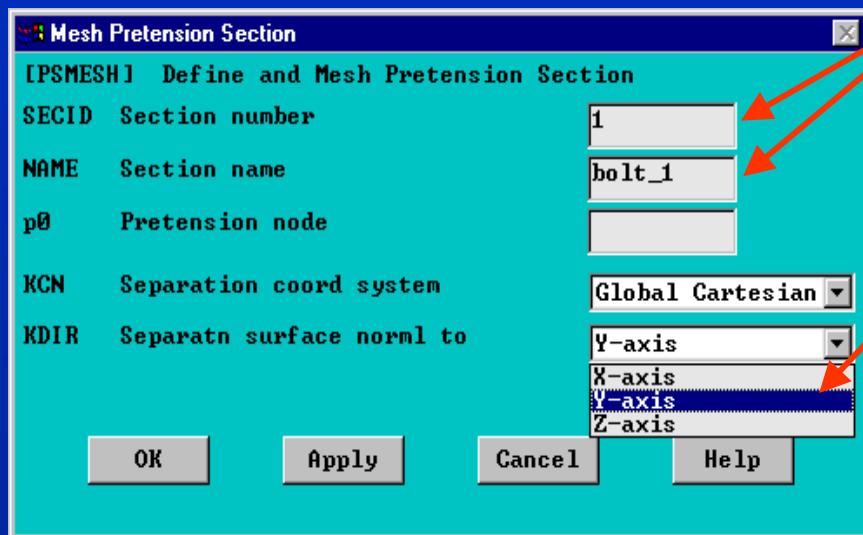
- Select the option to – Divide at Node – Picked Elements +
 - Follow the menu prompts



Typical Procedure

ANSYS

- Next, fill in the dialog box



Assign a section number and name

Specify the preload direction
(Y-axis points along the bolt axis in this case)

- Pressing OK will separate the elements of the bolt into two unconnected groups, tied together with pretension elements.

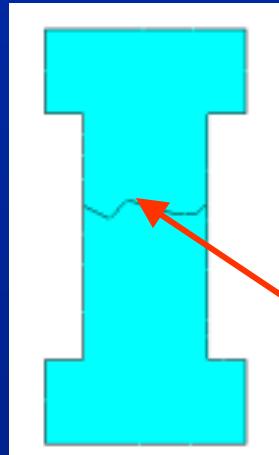
Typical Procedure

ANSYS

- The output window will list a summary of the elements and the pretension node number.

```
DEFINE AND MESH PRETENSION SECTION 1 bolt_1
CREATING A NEW PRETENSION NODE,
BY SEPARATING THE ELEMENTS OF AREA      2
AT COORDINATE SYSTEM    0, Y = Y OF NODE    162

CREATED PRETENSION NODE    548
AT GLOBAL X,Y,Z =      0.50000     0.0000     0.0000
CREATED ELEMENT TYPE    4 AS PRETS179
CREATED PRETENSION SECTION 1 bolt_1
WITH NORMAL DIRECTION  0.000000  1.000000  0.000000
CREATED    9 NEW PRETENSION ELEMENTS
MODIFIED MODAL CONNECTIVITY OF    4 EXISTING ELEMENT(S)
```

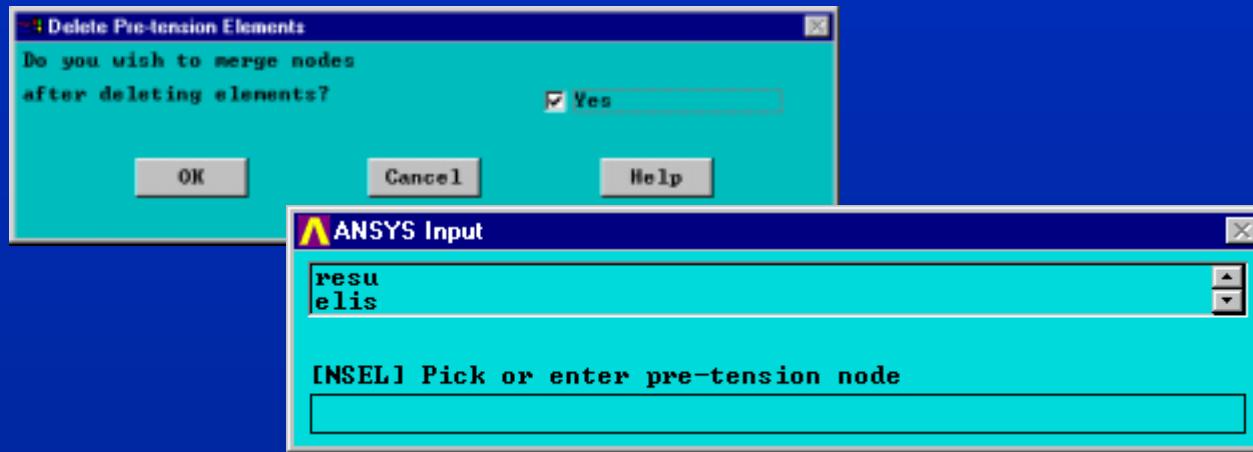


Notice how the Bolt is split
along existing element
boundaries

Typical Procedure

ANSYS

- You can “undo” a PSMESH operation as follows:
 - Preprocessor > Delete > Pre-tens Elements



- The GUI automatically deletes the pretension elements and merges the nodes at the pretension section.
 - However, the pretension node (K node) is not deleted.
 - Note: if you don't pick the pretension node, only individual pretension elements attached to that node will be deleted.

Typical Procedure

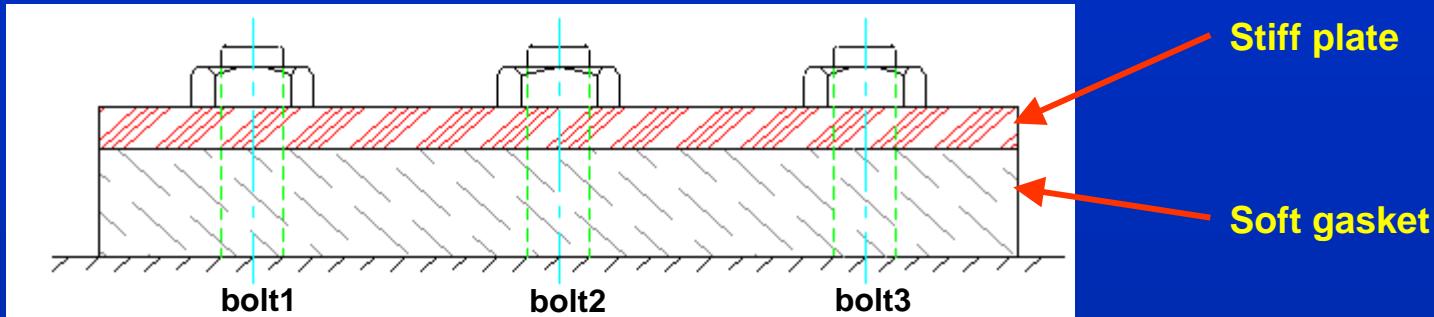


Pretension Load Application

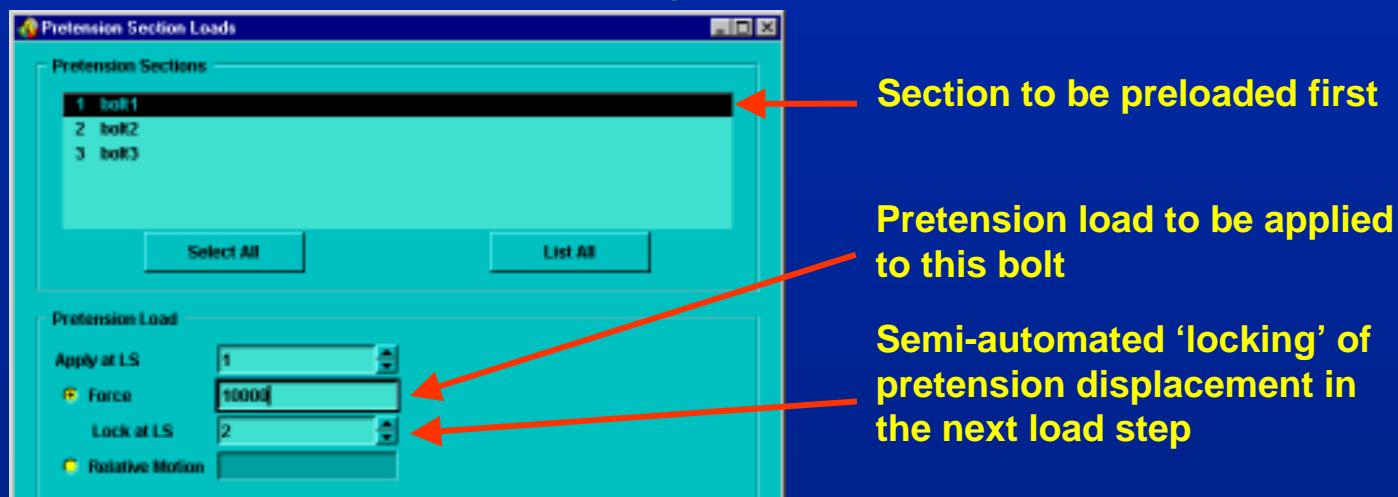
- When a physical bolt is pretensioned:
 - Turning the nut reduces the unstretched grip length of the bolt, thereby inducing pretension
 - When the desired pretension is achieved and the wrench is removed, the new unstretched grip length becomes ‘locked’
- Typical ANSYS pretension loading procedure represents this same sequence
 - First, apply the specified pretension (usually a specified force) in one load step
 - Then, lock the pretension section displacement (lock the shortened grip length) in a subsequent load step.
 - Once all bolts are pretensioned and locked, apply external loads in the final load step

Typical Procedure

ANSYS



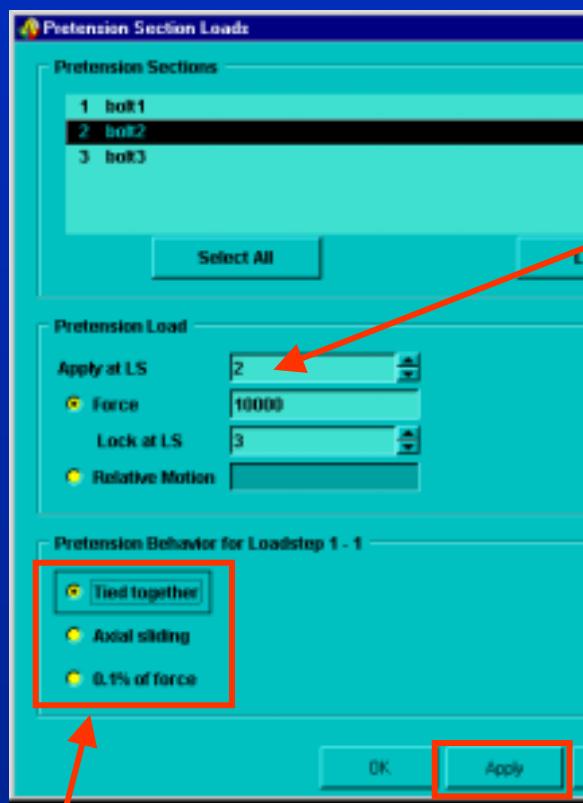
- Pretensioning and locking a large number of bolts in a specified sequence is made easy with the load-management tool.
 - Solution > Loads > Apply > Pretnsn Sectn ...



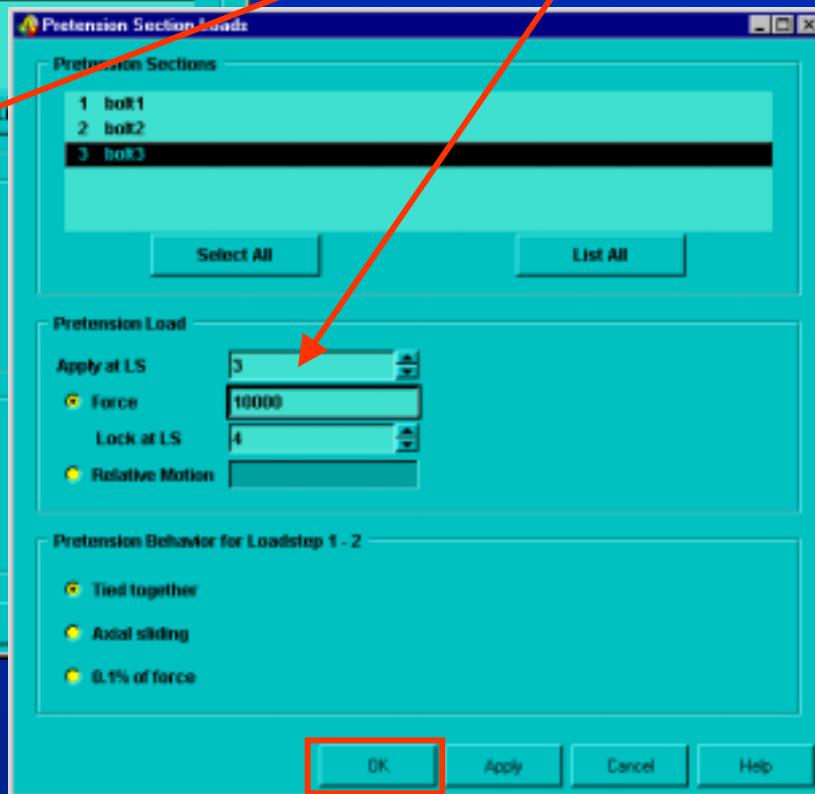
Typical Procedure

ANSYS

- Continue the loading sequence for the other two sections:



Semi-automatic restraint of the free-body bolt pieces



Incrementing the loadstep number controls the pretensioning sequence

Typical Procedure

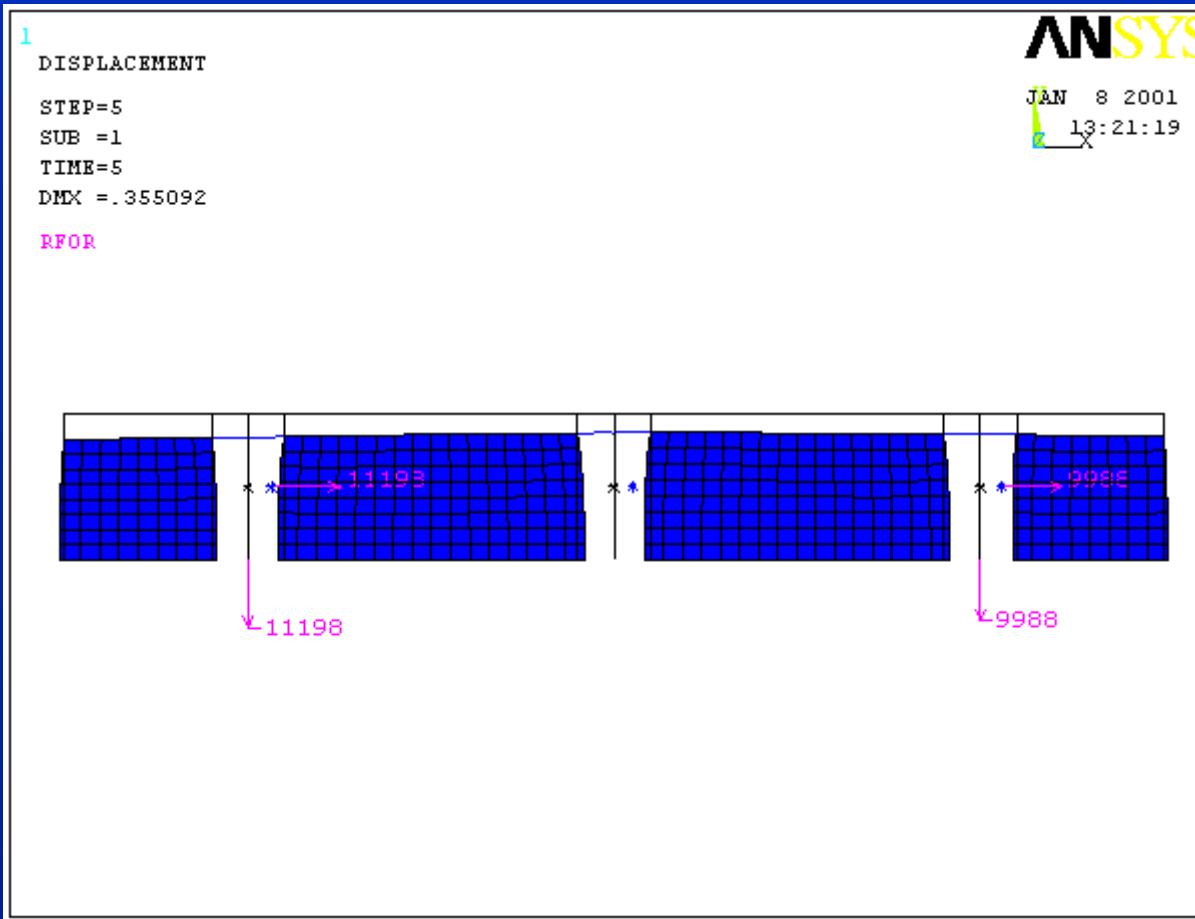


- In solution, ANSYS automatically applies and locks the loads in the specified load step every time you issue a SOLVE.
 - For the three bolts shown in the previous example, you would need to solve four times to apply the specified pretension sequence.
 - SOLVE (LS1, applies the preload to bolt1)
 - SOLVE (LS2, fixes the preload displacement for bolt 1 and tightens bolt2)
 - SOLVE (LS3, locks bolt 2 and tightens bolt3)
 - SOLVE (LS4, locks bolt3)
- With the preload applied, apply external loads acting on the overall structure.

Typical Procedure

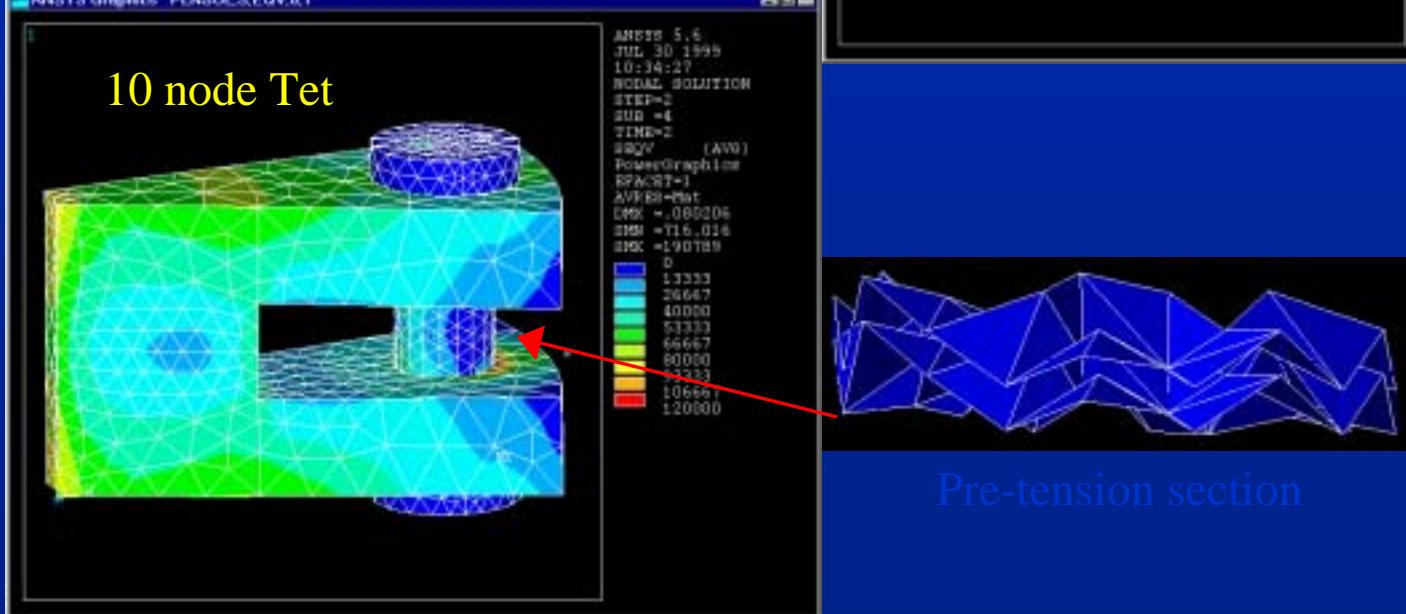
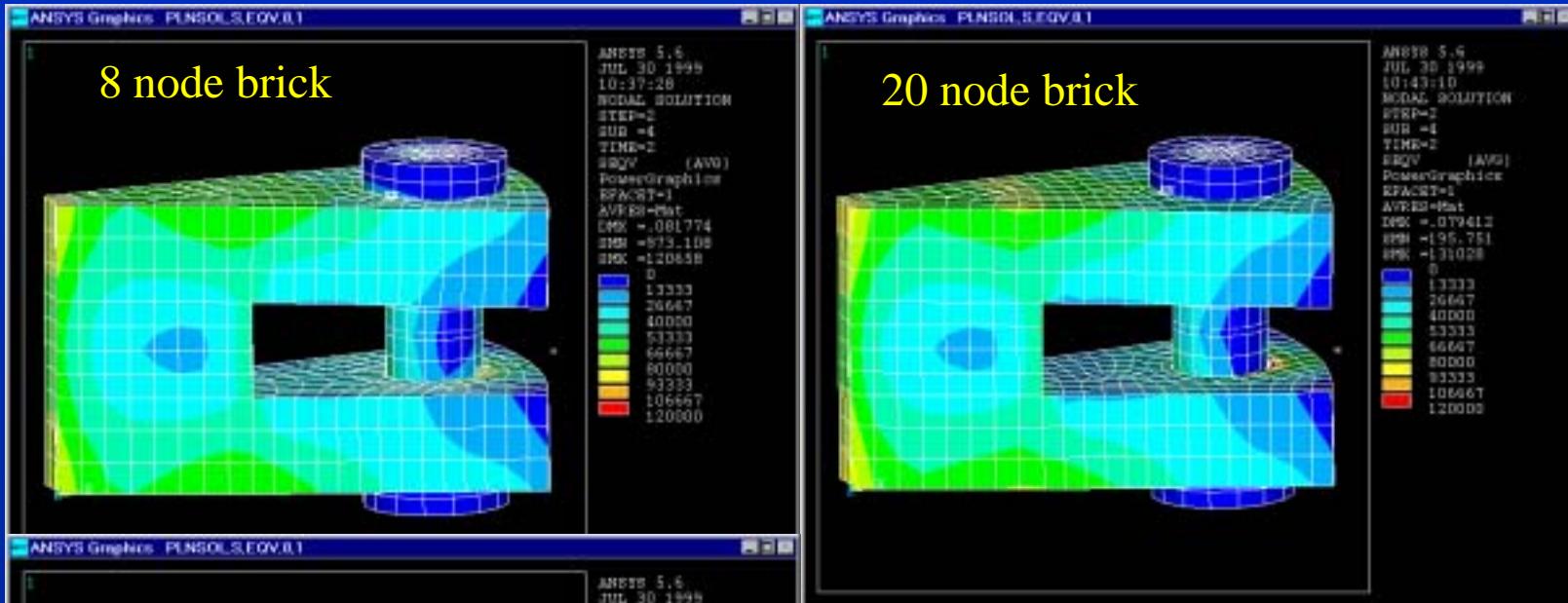
ANSYS

- Deformed geometry plot and bolt forces for the previous example



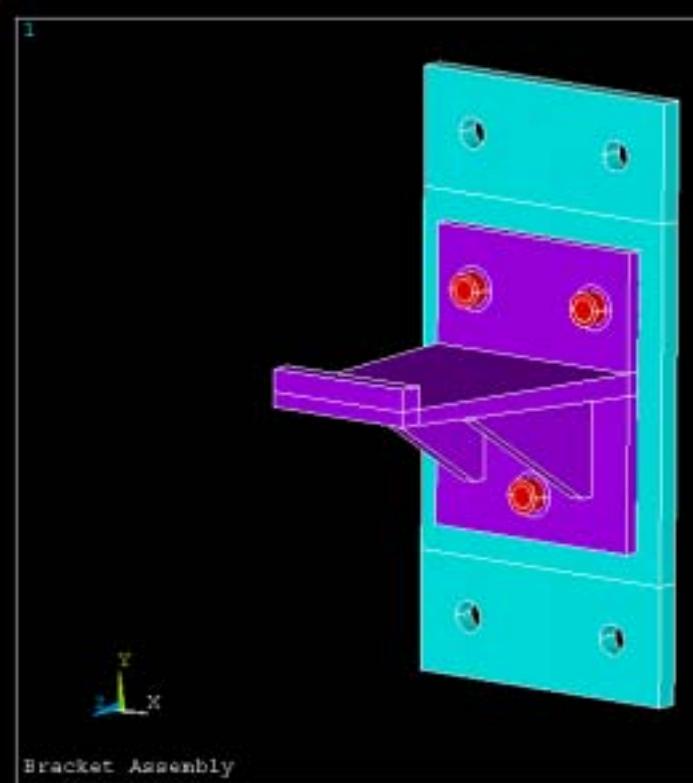
Application: 3D bolt Joint simulation

ANSYS



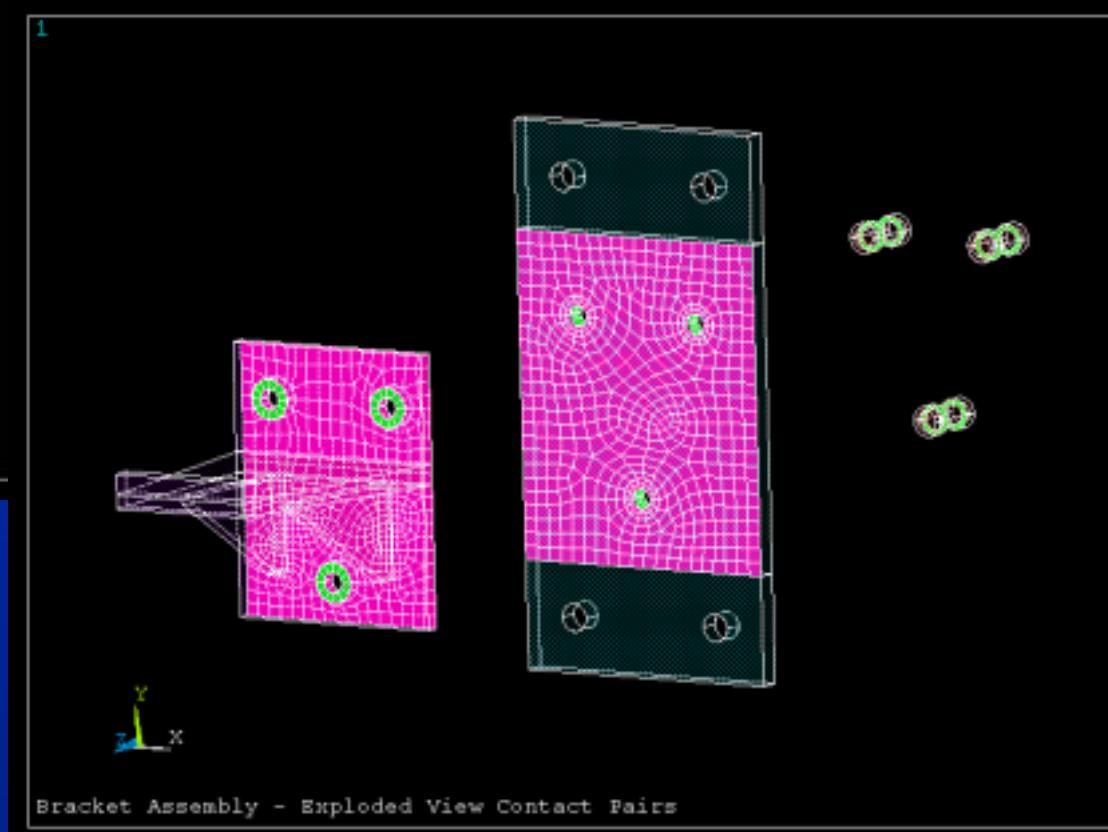
Application: Bracket Assembly

ANSYS



ANSYS

Contact Pairs

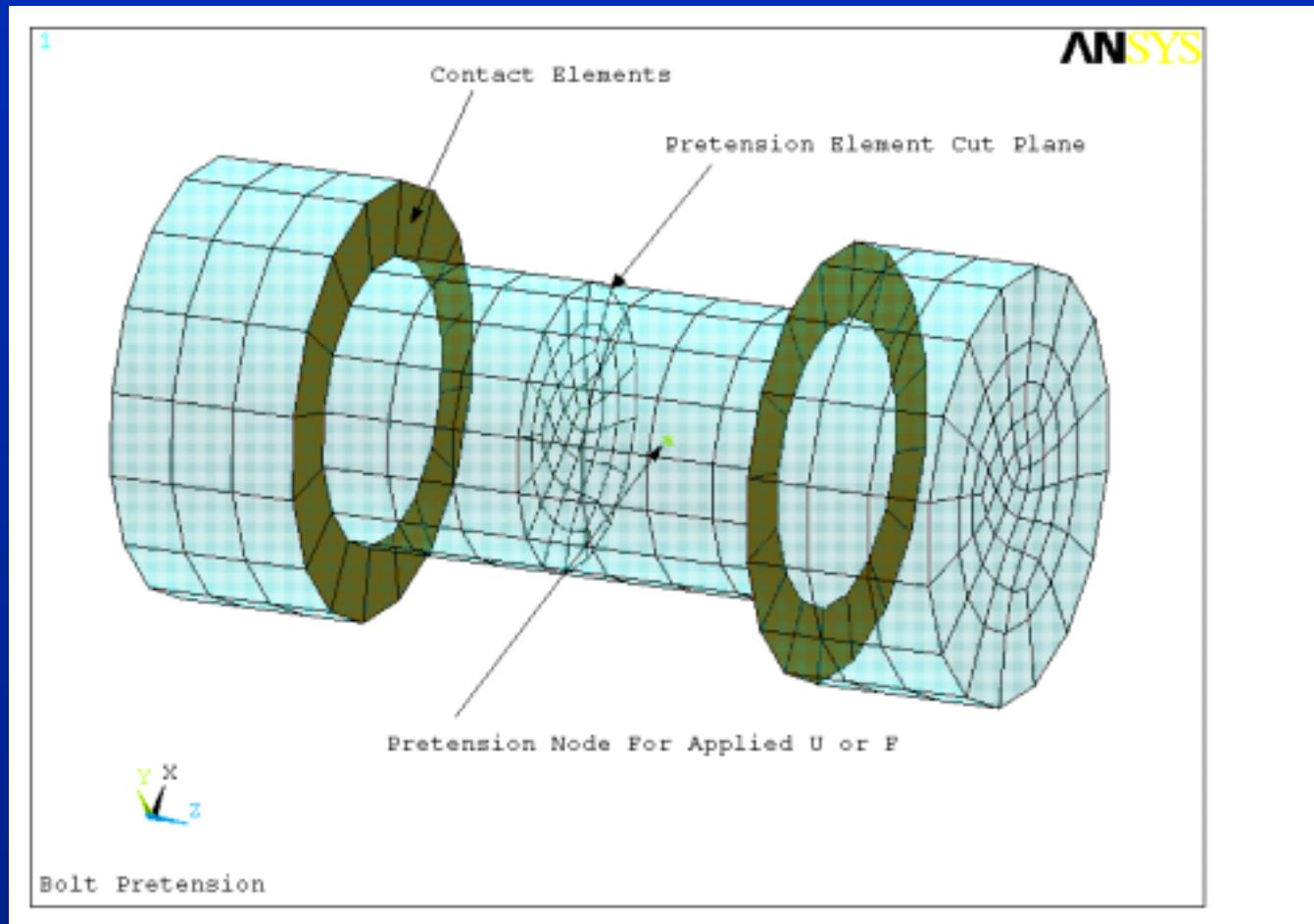


Application: Bracket Assembly

ANSYS

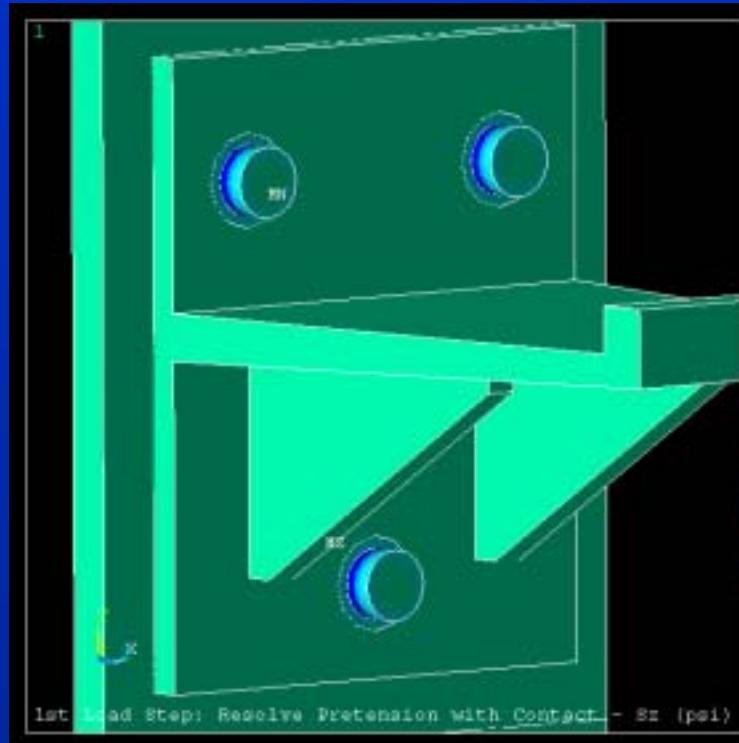
Static Analysis: Loading

First load step: Resolve bolt pre-tension ($F_i=5,000$ lb)

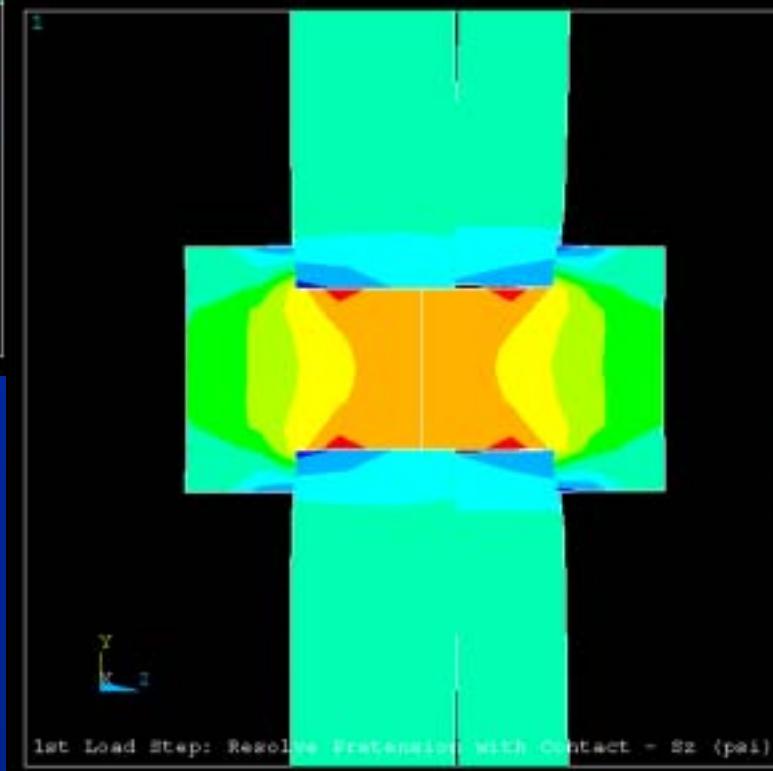


Application: Bracket Assembly

ANSYS



```
ANSYS 5.6.1
MAY 19 2000
09:19:21
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
S1      (AVG)
R8Y8=0
PowerGraphics
EFACET=1
AVRE8=Mat
DMX = .002305
SMN = -46726
SMX = 89462
-46726
```

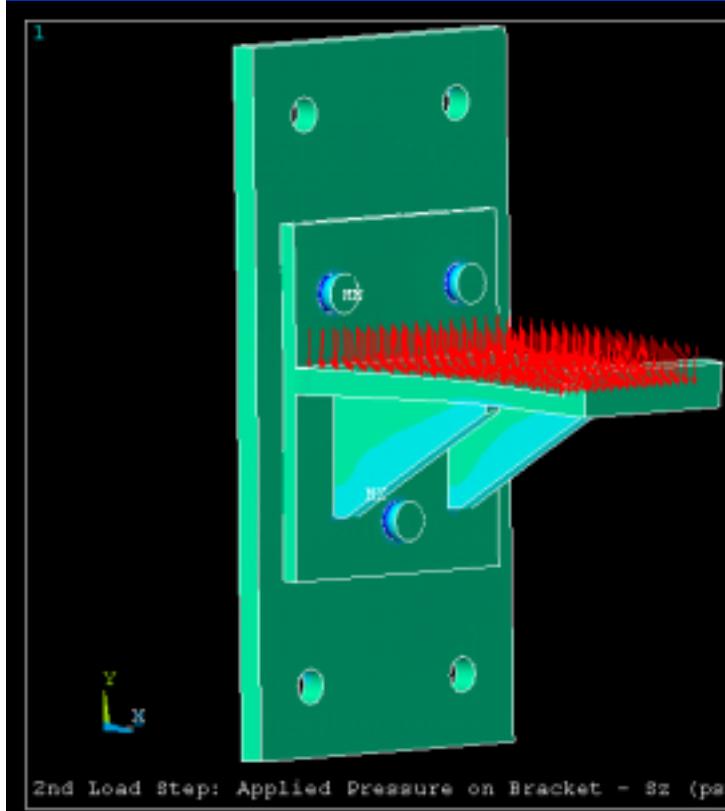


```
ANSYS 5.6.1
MAY 19 2000
09:21:35
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
S2      (AVG)
R8Y8=0
PowerGraphics
EFACET=1
AVRE8=Mat
DMX = .001322
SMN = -46726
SMX = 64401
-46726
-34378
-22031
-9603
2664
15011
27359
39706
52054
64401
```

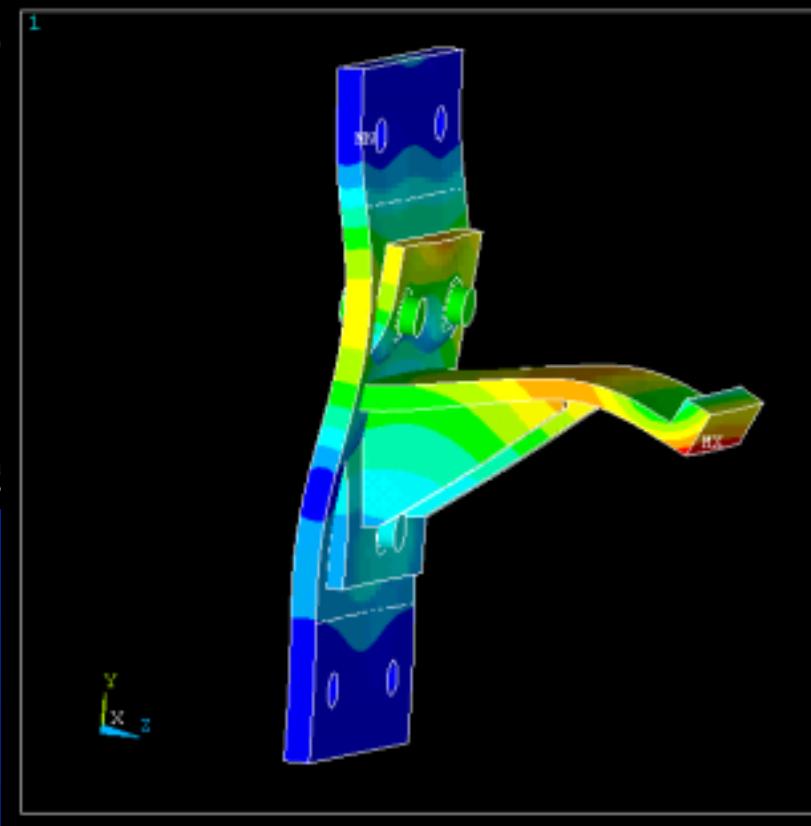
Results for First Load Step

Application: Bracket Assembly

ANSYS



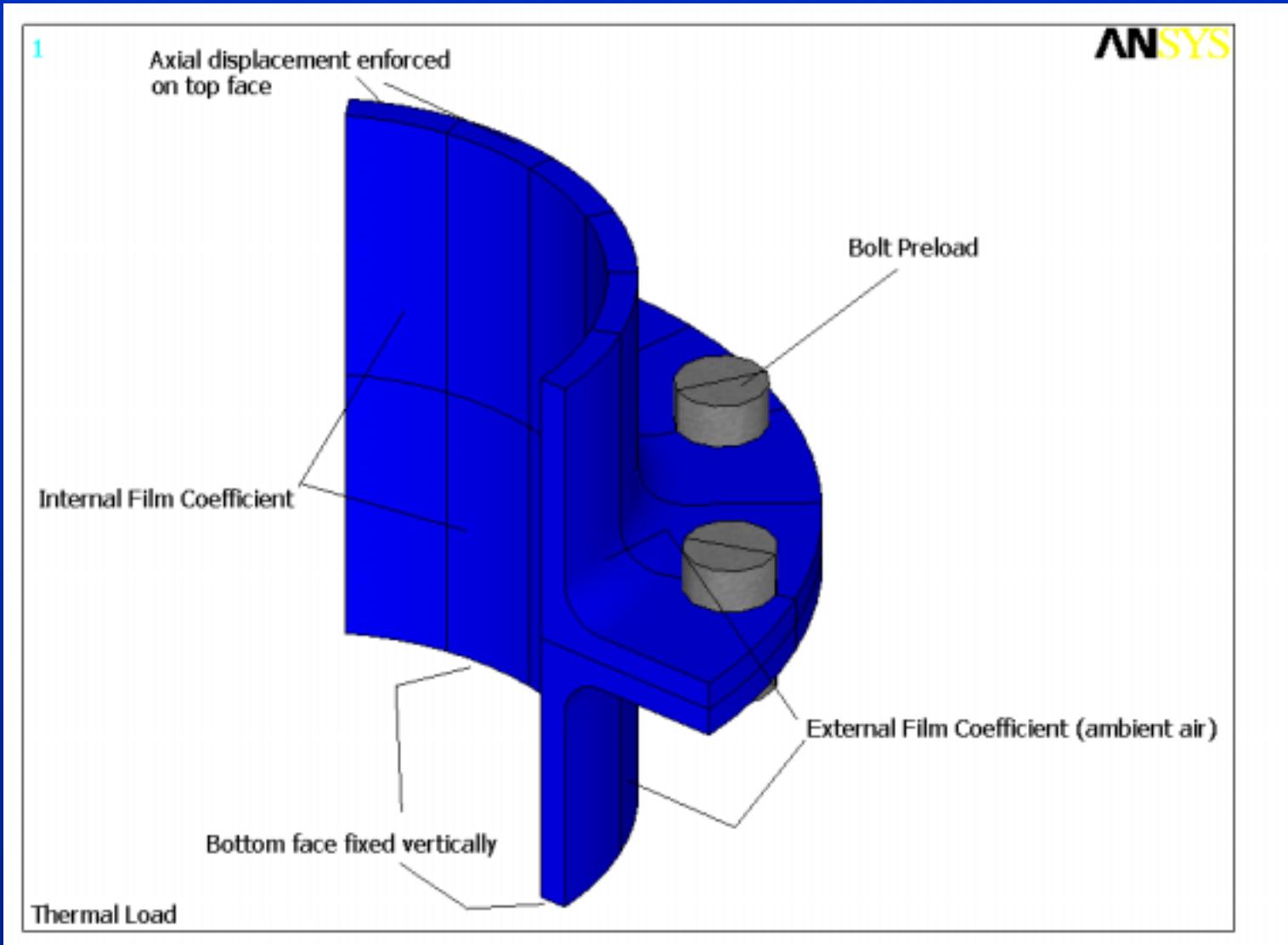
Modal analysis
Contact status maintained



Second Load Step
Applied pressure on bracket
(1,500 lb total load)

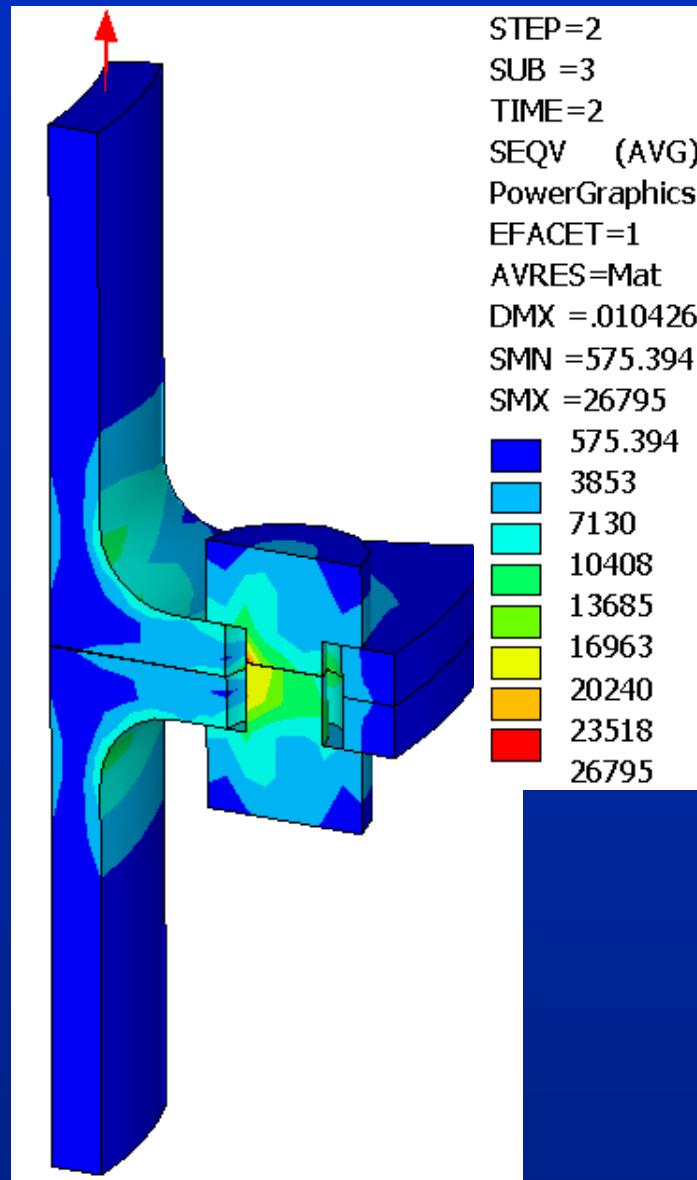
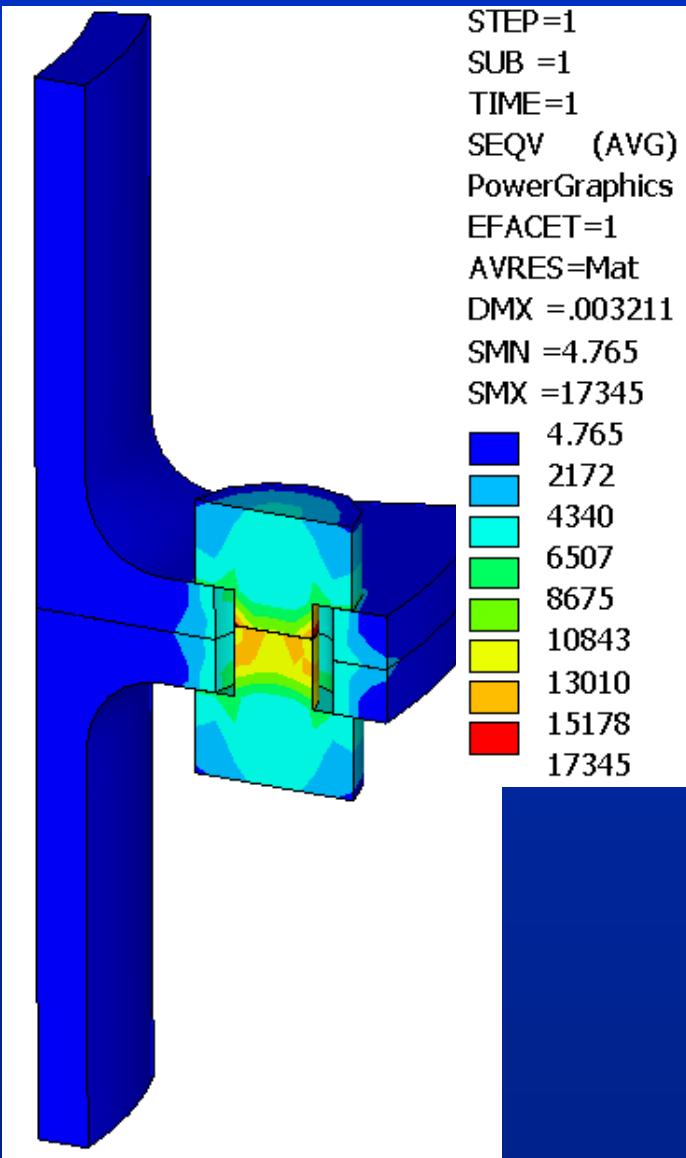
Application: Bolted Joint Thermal Contact

ANSYS



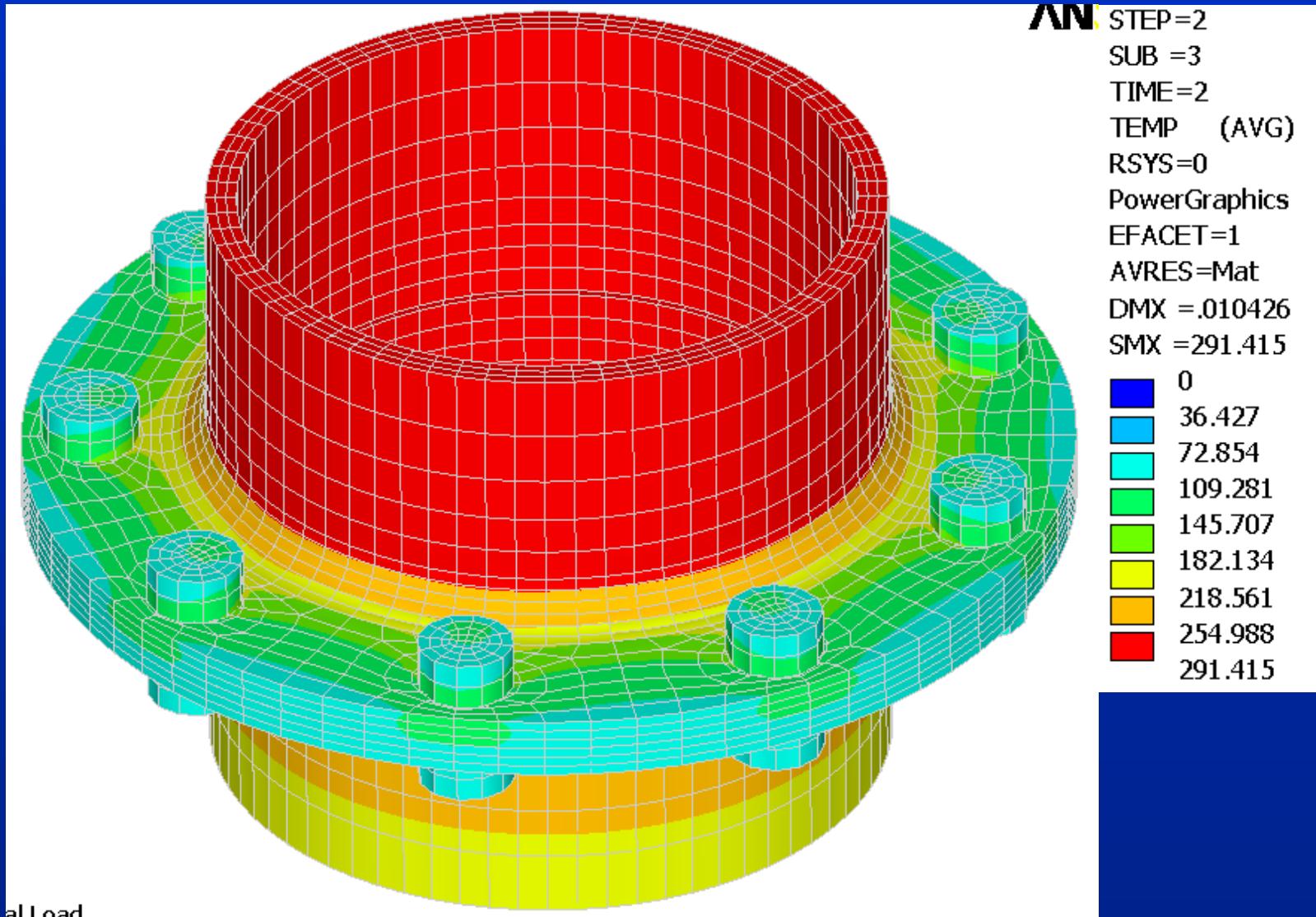
Application: Bolted Joint Thermal Contact

ANSYS



Application: Bolted Joint Thermal Contact

ANSYS



Solver Issue for Contact Analysis

Solver issue

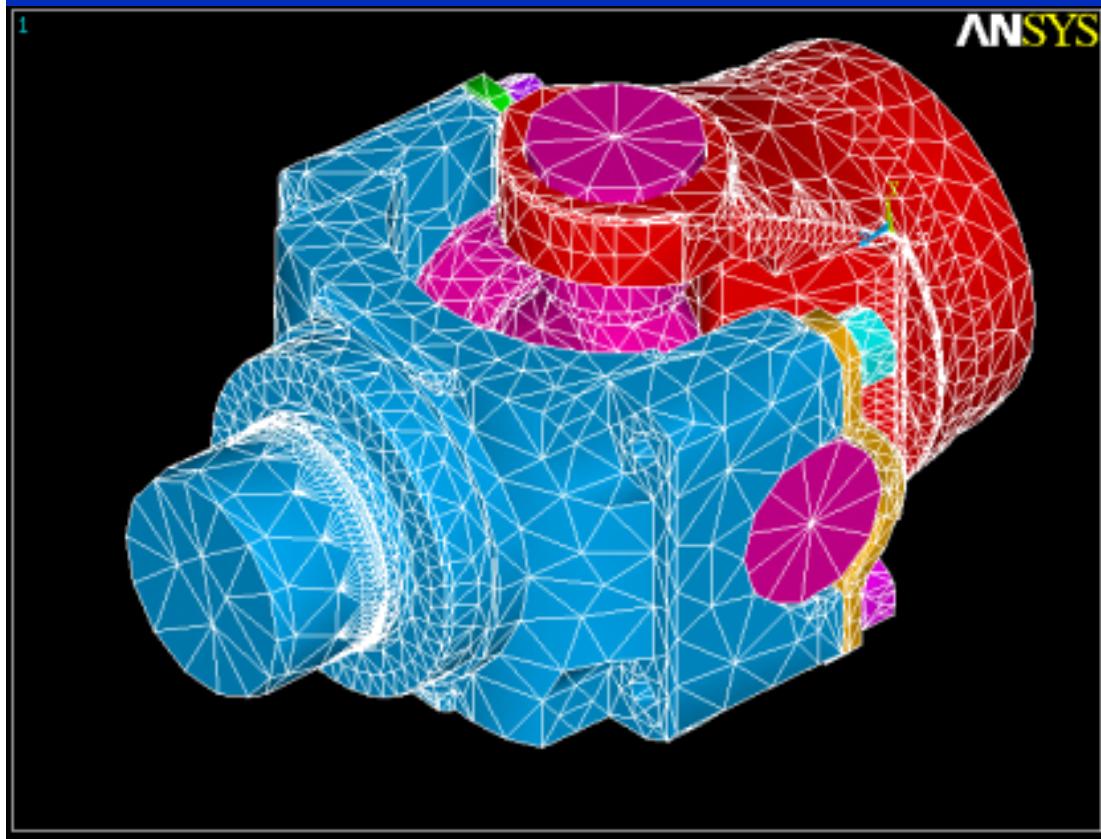


- **Sparse solver is the default solver**
 - Best for medium contact models (under 1,000,000 DOF)
 - Good for slender/thin structure
 - Direct solver - more robust than PCG solver
 - Unsymmetric system is available but expensive
- **PCG solver as an option**
 - Enhanced for handling indefinite matrix
 - Large contact models (over 1,000,000 DOF)
 - Good for bulky solid
 - Iteration solver - performance bases on element shape and contact condition
 - Not applicable for pure Lagrange algorithm, unsymmetric solver

Solution Options



- Sparse solver is the default solver



Total DOF = 502851

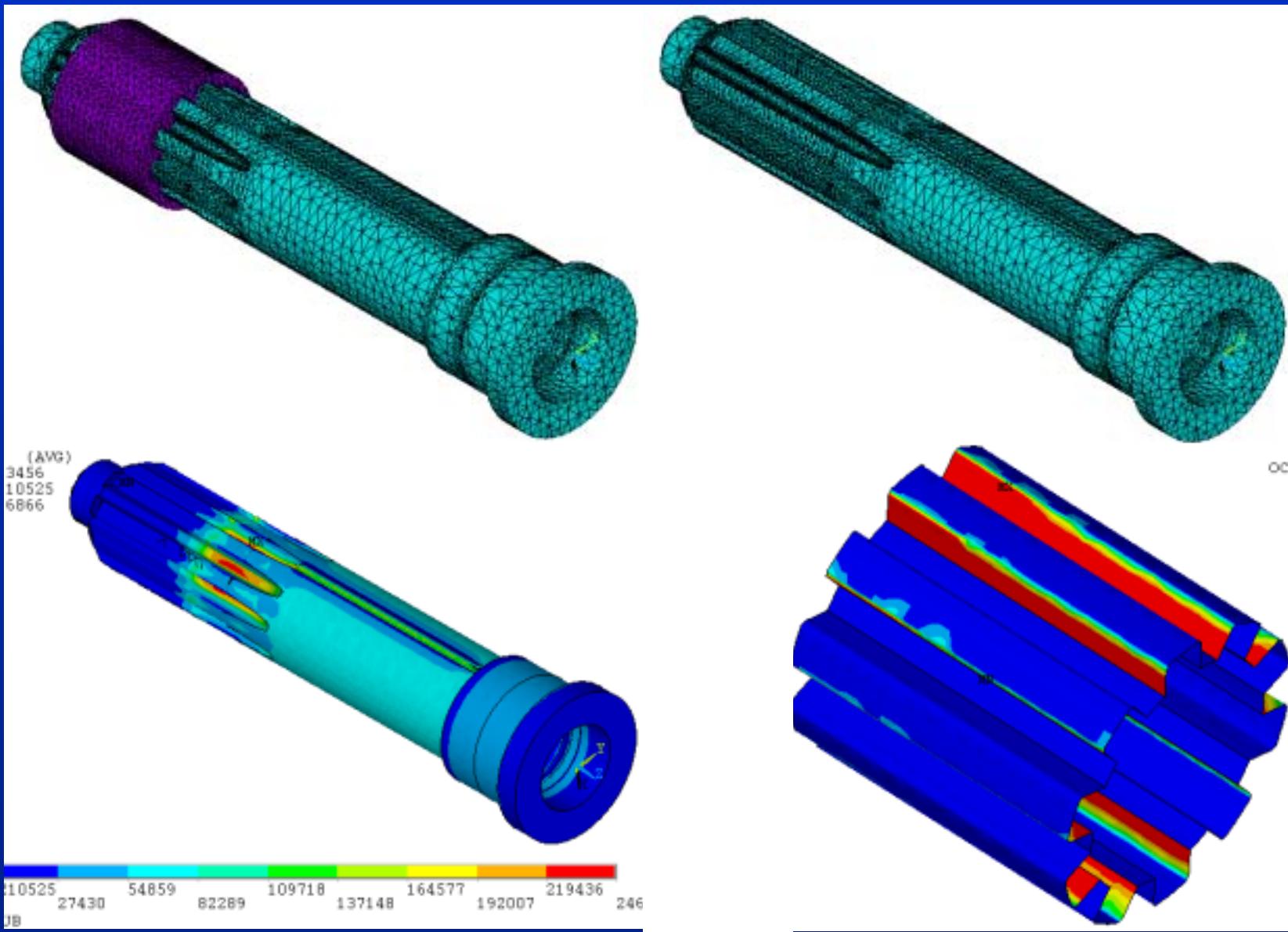
**# Sparse in ansys57
Memory=480, CPU=2123**

**# Sparse in ansys60:
Memory=312, CPU=1146**

**# PCG
Memory = 294, CPU=1014**

Application: Splined Shaft

ANSYS



Application: Splined Shaft

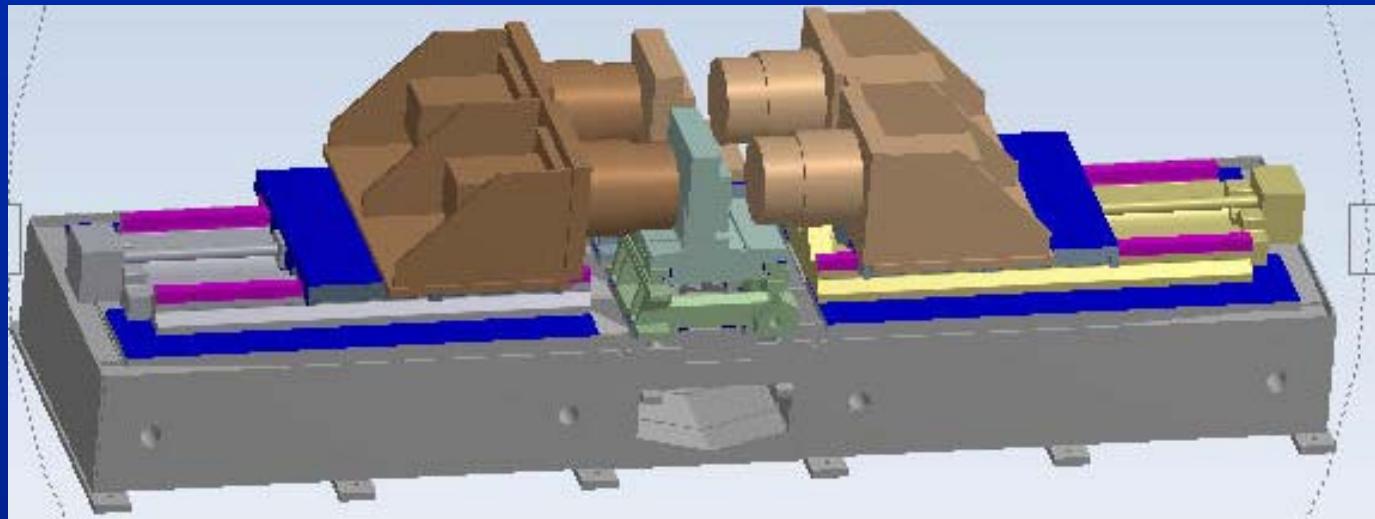


```
Number of equations = 497380,      Maximum waveform =      383
EQUIL ITER    1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.4047E-01
  FORCE CONVERGENCE VALUE = 0.1941E+05  CRITERION= 18.79
EQUIL ITER    2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.7171E-03
  FORCE CONVERGENCE VALUE = 0.3028E+05  CRITERION= 16.13
EQUIL ITER    3 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.3307E-03
  FORCE CONVERGENCE VALUE = 108.6          CRITERION= 16.56
EQUIL ITER    4 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.5784E-04
  FORCE CONVERGENCE VALUE = 9.115          CRITERION= 16.88      <<< CONVERGED
CPU TIME SPENT FOR CONTACT DATABASE           34.478 = 1.2 %
                                              CONTACT SEARCH       181.013 = 6.4 % = 8.12 %
                                              CONTACT ELEMENTS   14.210 = 0.5 %
PCG SOLVER          OTHER ELEMENTS        762.809          Time consumed
                                              EQUATION SOLVER    1869.768
                                              TOTAL SYSTEM       2827.800
SPARSE SOLVER        TOTAL SYSTEM       5920.107
```

Solution Options

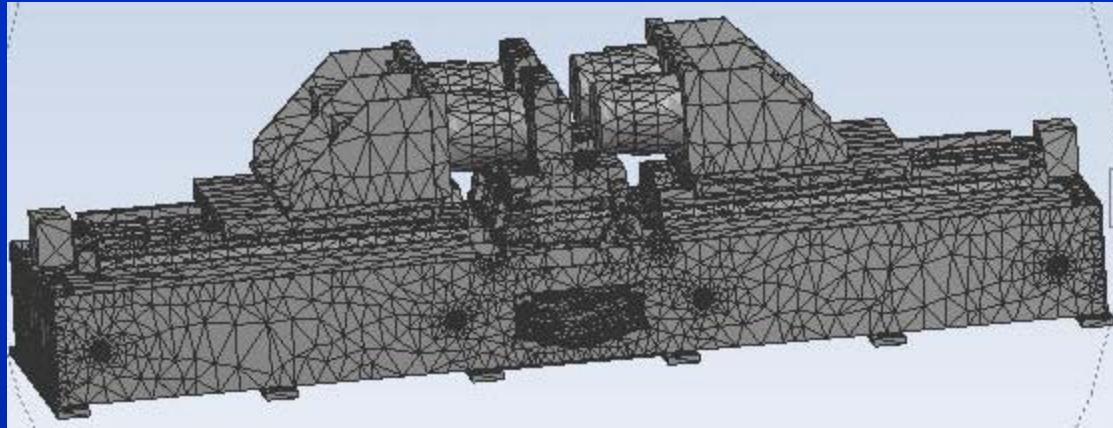
ANSYS

- **AMG Algebraic multigrid solver even better**
 - **Insensitive to matrix ill-conditioning (i.e. high aspect ratio elements, contact)**
 - **Scalable up to 8 processors (shared- memory only)**
 - **30% more memory required than PCG**



3D Assembly

ANSYS



Finite Element Model
119,000 Elements
590,000 DOFs

Method	Memory Mbytes	Iter 10^{-6}	Solver Elapsed Time (sec)			
			NP=1	NP=2	NP=3	NP=4
PCG	300	5301	8679	7745	6636	6909
AMG	722	200	5265	3831	2638	1884

AMG shows superior convergence and scaling for this problem

Tips and Troubleshooting for Contact Analysis - A Diagnostic tools

Non contact-related issues



- **Unrealistic physical model**
- **Unreasonable loading and boundary conditions**
- **Poorly/coarsely discretized mesh, sharp corner, location of mid-side nodes, element distortion**
- **Hourglass and locking**
- **Unreasonable or incorrect material properties and bad input unit**
- **Large plastic deformation**
- **Incompressible or near incompressible**
- **Local & global instabilities**
- **Follower loads**

What You Should Do in the Begin

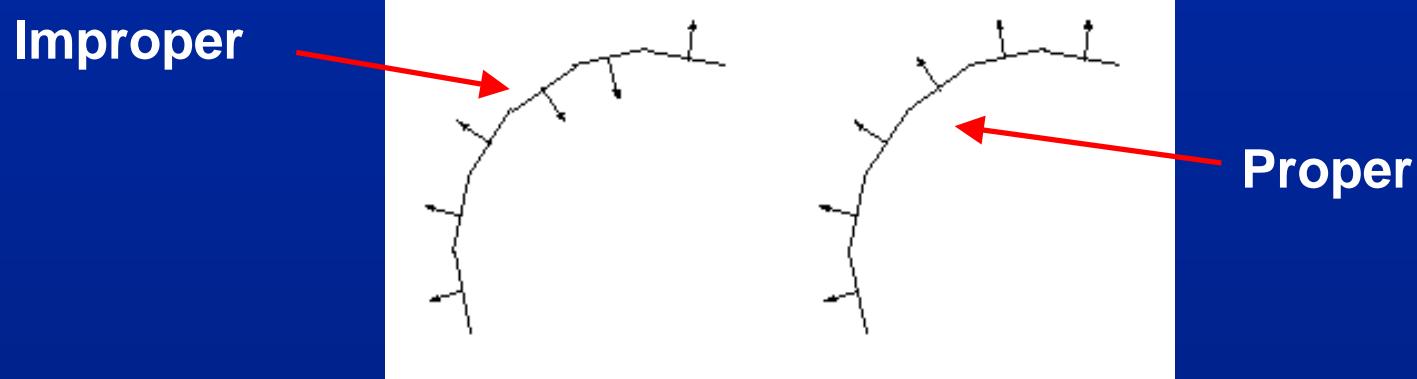


- Read Chapter 10 of Ansys structural analysis guide
- Verify outward normal of contact/target surfaces
- Verify pinball region
- Verify constraints
- Verify initial contact status through output
 - Issue: PSOLVE, elform
- Use CONCNTR, PRINT, level, to get detailed printout of the contact state in the output file so that you can diagnose the problem.

What You Should Do in the Begin: Tip

ANSYS

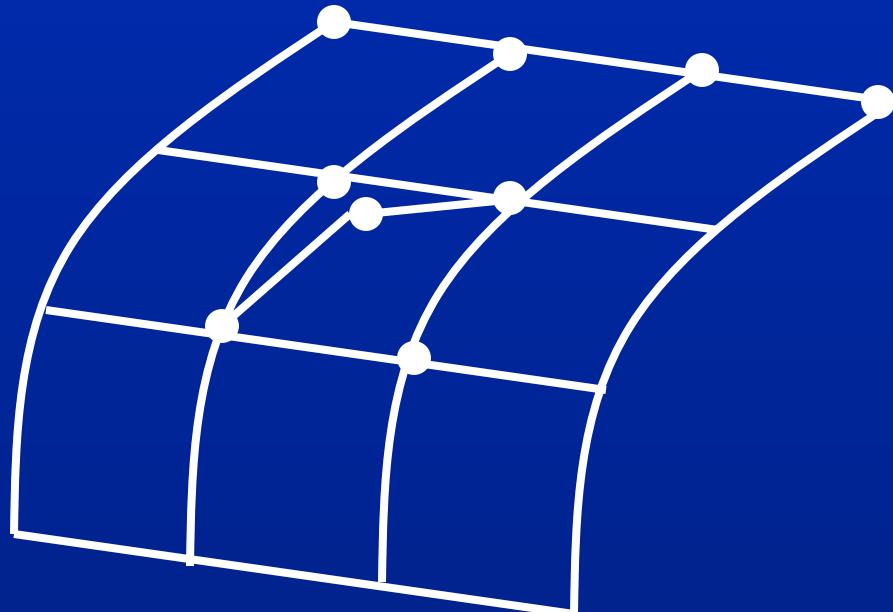
- Rigid-to-Flexible and Flexible-to-Flexible contact can be defined in the same model. Be sure to use separate real constants to define the contact pairs. The output file information can be helpful to check your contact pairs.
- Be sure to check the outward normal direction of the contact and target elements (via Main Menu > Preprocessor > Create > Contact Pair > View and Edit...) Contact occurs on the positive outward normal side of the contact and target elements.



What You Should Do in the Begin: Tip

ANSYS

- Defining duplicate nodes contact/target surface
 - On contact surface
 - Overconstraints if using pure Lagrange mutiplier
 - On target surface
 - Contact node may stuck there



What You Should Do in the Begin: Tip

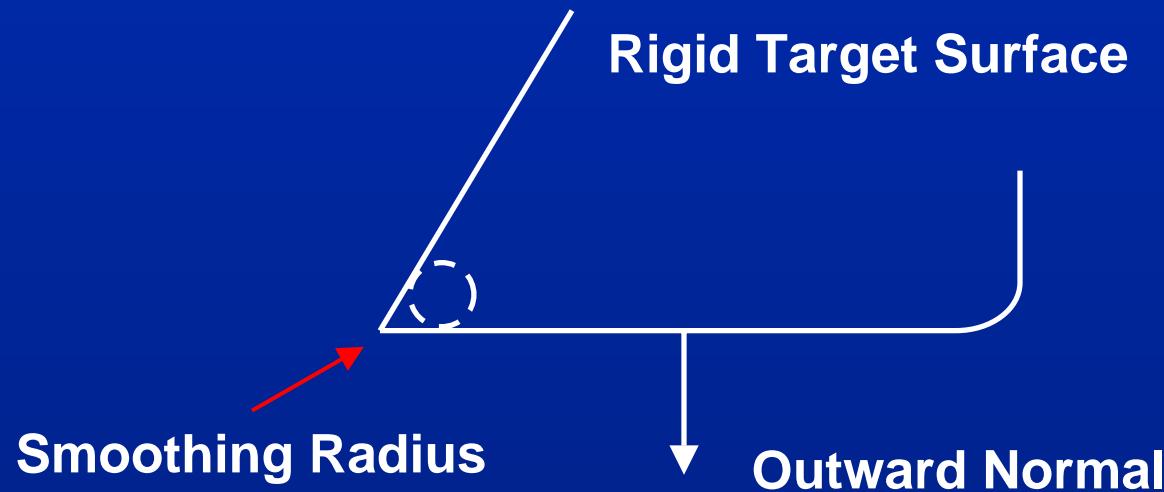


- **Using poorly discretized surfaces**
 - Unrealistic penetration occurs with coarsely discretized contact surface. Use refiner mesh to define contact accurately.
 - Coarsely discretized, curved target surfaces can lead to unacceptable solution accuracy. Using a more refined mesh or higher order mesh will improve the overall accuracy of the solution.
- **Ensure that the target and contact surface definitions extend far enough to cover the full expected range of motion for the analysis.**

What You Should Do in the Begin: Tip

ANSYS

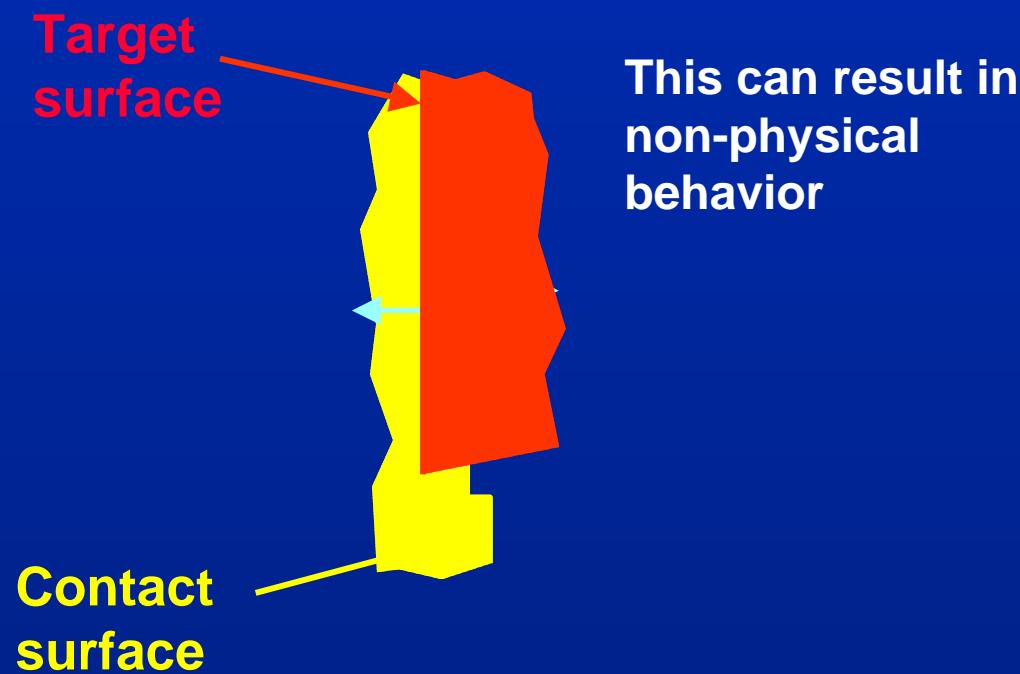
- Sharp corners on the contact/target surfaces:
 - Smooth the surfaces.
 - Use refined mesh around corner.
 - Use unsymmetric solver.
 - If the physical problem has a sharp concave fold, use two separate contact pair definitions.



What You Should Do in the Begin: Tip

ANSYS

- Interference fit problems with widely varying contact normal directions can result in non-physical behavior. In this case try switching the target and contact surfaces. Or, try to use contact normal being perpendicular to target surface (KEYOP(4)=2)



Output File Information



- Summary of contact status for each contact pair
 - CONCNTR,print, -1 – No contact pair information
 - CONCNTR,print, 0 - Limited information
 - At beginning of solution
 - At the stage of solution divergence
 - CONCNTR, print, 1
 - At the end of each sub-step
 - CONCNTR, print, 2
 - At the end of each iteration
 - CONCNTR, print, 3
 - For individual contact point

More
Info



Output: CONTACT INFORMATION PRINTOUT LEVEL 3

Output File Information

- The output file will also echo the initial contact penetration/gap selected for each contact pair.
- Any warnings and errors encountered in the analysis phase are written to the output file. You must look in the file to find the reasons why the analysis stopped prematurely.

*** NOTE ***

CP= 0.000 TIME= 00:00:00

Rigid-deformable contact pair identified by real constant set 2 and contact element type 3 has been set up. Please verify constraints on target nodes which may be automatically fixed by ANSYS.

Contact algorithm: Augmented Lagrange method

Contact detection at: nodal point (normal from contact nodes)

Default contact stiffness factor FKN	1.0000
The resulting contact stiffness	2579.8
Default penetration tolerance factor FTOLN	0.10000
The resulting penetration tolerance	0.22786E-01
Max. initial friction coefficient MU	2.1000
Default tangent stiffness factor FKT	1.0000
Default Max. friction stress TAUMAX	0.10000E+21
Average contact surface length	0.34860
Average contact pair depth	0.22786
Default pinball region factor PINB	3.0000
The resulting pinball region	0.68358
Default target edge extension factor TOLS	2.0000
Initial contact closure factor ICONT	0.10000E-07
The resulting initial contact closure	0.22786E-08

WARNING: Initial penetration is included.

*** NOTE ***

CP= 0.000 TIME= 00:00:00

Max. Initial penetration 0 was detected between contact element 454 and target element 435.

*** WARNING ***

CP= 0.000 TIME= 00:00:00

Max. Friction coef. 2.1 has defined in the model. Switch the unsymmetric solver (NROP,UNSYM) instead if convergence difficulty is encountered.

ANSYS

Too big penetration at the Beginning of the Analysis



- **Read output to see the contact penetration pair based.**
- **Real initial interference**
 - Use Ramped option KEYOP(9)=2,4 solving initial interference over several sub-steps.
 - For reasonable small one use KEYOP(9)=1
 - Use smaller FKN and increase FKN in subsequent load steps
- **Spurious contact:**
 - Redefine PINB
 - Check surface normal & flip the surface normal

Troubleshooting

- Very small displacement correction, but the tolerance on residual force is not satisfied due to:
 - Contact length too small
 - Warning: Min. contact depth 5.376d-6 is too small which may cause accuracy problem, you may scale the length unit in the model.
 - The Elastic moduli or force/mass quantities too big
 - Warning: Max. contact stiffness 1.21d16 is too big which may cause accuracy problem, you may scale the force unit in the model
 - Contact surface offset using CNOF or Bonded always
 - Introduce rotation energy
 - Contact stiffnesses are too large
 - Solution: scale units or modify CNVTOL or reduce FKN, FKT, or use PSOLVE,

Troubleshooting (Pair Based)

SUMMARY FOR CONTACT PAIR IDENTIFIED BY REAL CONSTANT SET 2

Max. Penetration of -7.127220673E-04 has been detected between contact element 454 and target element 435.

If none of pair in contact, rigid body motion can occur

*** NOTE *** CP= 0.000 TIME= 00:00:00

Contact element 454 has Max. Slip distance 7.618911816E-04.

If the slip dis. is too big, increase FKT

*** NOTE *** CP= 30.360 TIME= 18:41:35

8 contact points have too much penetration (do not meet compatibility condition)

Contact stiffness FKN is too small? Increase FKN

For initial penetration: ignore it by keyopt(9)=1 or ramped options 2,4

Contact normal Points wrong direction.

Spurious contact: reduce the pinb

Troubleshooting (Pair Based)



*** NOTE *** CP= 0.000 TIME= 00:00:00

6 contact points have abrupt change in contact status.

Load step size is too big: use auto time prediction keyopt(7)>0

Pinball is too small, increase PINB

The contact node is sliding off the target edge: increase TOLS

*** NOTE *** CP= 0.000 TIME= 00:00:00

Contact element 454 has the highest chattering level 7.

Contact stiffness is too big: reduce FKN, FKT

Load step size is too big

Sharp corner or coarse mesh: refine mesh

Stick-sliding: use NROP,unsym?

The contact node is sliding off the target edge: increase TOLS

Divergence of Solution



- **Too Many Cutbacks in the Last Time Increment**
 - Restart the analysis
 - Reset solution options, number of equilibrium iterations
 - Increase the number of allowable increments on the NSUB option and redo the analysis.
- **Failure to determine the contact state change occurs**
 - in first increment: Possible causes include too big the penetartion
 - Occurs after the first increment: Possible causes are contact chattering
- **Failure to achieve equilibrium: force residuals**

Failure to achieve equilibrium: force residuals



- Unconstrained rigid body motion indicated by the presence of very large displacement corrections.
- Overconstraints usually indicated by zero pivot/negative warning messages.
- Buckling indicated by the load-displacement curve reaching a load maximum: use arc-length algorithm.
- Friction in the model

Friction in the Model

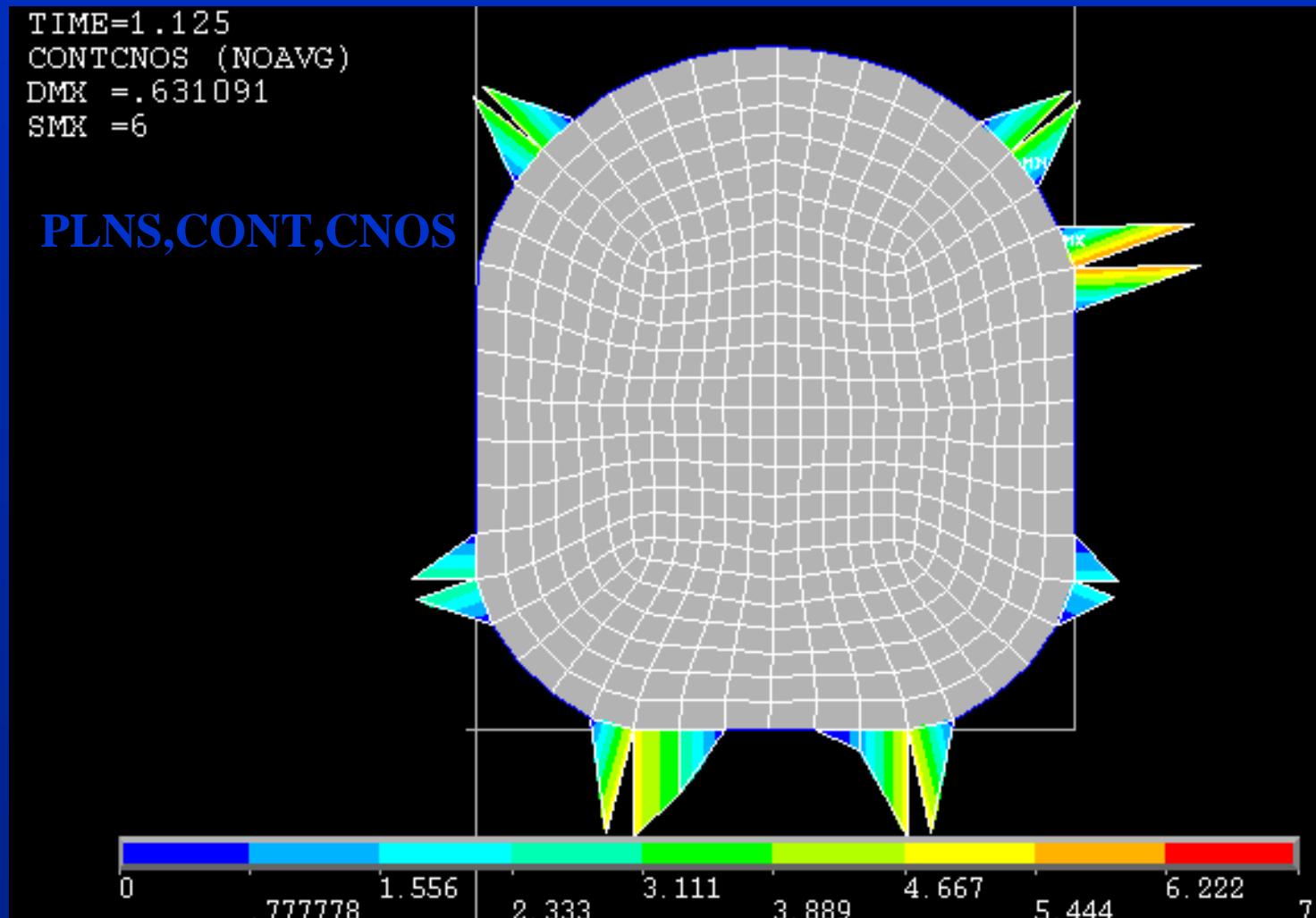


- Use frictionless contact or rough contact to see any help
- If the difficulty is indeed due to friction, then consider the following:
 - Use smaller tangent stiffness.
 - Use the unsymmetric solver (NROP,unsym) for curved target surface even if the coefficient of friction is small.
 - Re-examine the choice of the friction coefficient.
 - Refine the mesh so that more points come into contact at the same time.
 - Use bonded contact instead of rough to prevent convergence problems.

Contact Chattering

ANSYS

- Add chattering measurement CNOS to PLES, PLNS, PRES, PRNS



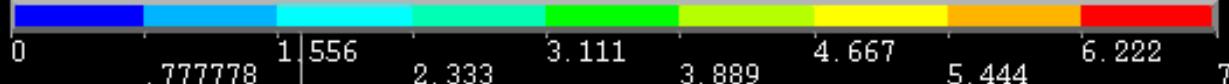
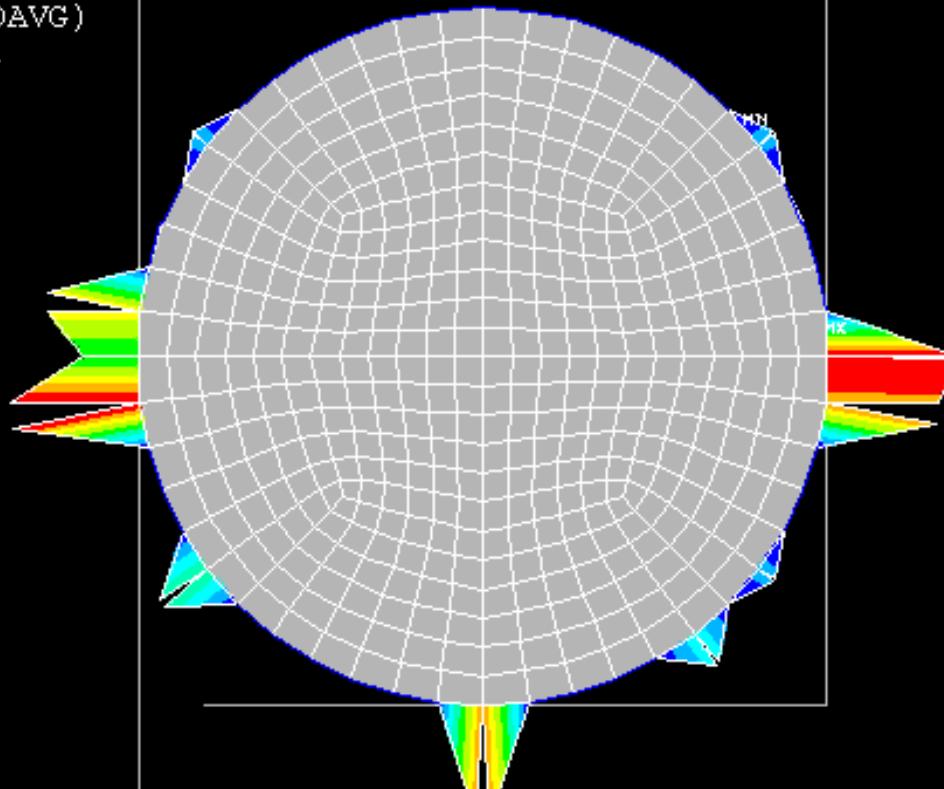
Contact Chattering

ANSYS

```
1 ELEMENT SOLUTION  
STEP=1  
SUB =1  
TIME=.1  
CONTCNOS (NOAVG)  
DMX = .063001  
SMX =7
```

ANSYS

JUN 5 2002
15:58:22



Causes of Contact Chattering



- **Contact stiffnesses are too higher: reduce the initial stiffness or redefine during load step.**
 - Often occur when the model has long, flexible parts with small contact pressures.
 - FKN and FTOLN need to be set appropriately: FKN usually will be between 0.01 and 10. Use a value of 1.0 (default) for bulk deformation problems, and 0.1 for bending dominated problems. Do *NOT* set FTOLN too small, too tight a penetration tolerance can result in divergence.
- **Only a few nodes are in contact: refine the underlying mesh of the contact surface or reduce the contact stiffnesses to distribute the contact over more nodes.**
- **The size of the region in contact is rapidly changing: use keyopt(7) to control time increment.**
- **The target surface is not sufficiently smooth : smooth the target surface by refining the underlying mesh.**

Causes of Contact Chattering



- A contact node is sliding off the target surface:
 - Ensure that the target and contact surface definitions extend far enough to cover the full expected range of motion for the analysis.
 - When modeling contact, ensure that the target surface edge is extended far enough for all expected motions of the contacting parts of the model. Use CONCNTR,print,3 option to monitor the history of the contact node that might slide off the target surface and find where the target surface needs to be extended more.
- *For extremely difficult situations*, you can allow some points to violate contact conditions by setting the numb on the CONCNTR, MAXP option. These parameters can be reset in a subsequent step.
- *If none of the above cases seem to apply*, try using the transient option

Troubleshooting (Solution)



Allow some points to violate contact conditions (too much penetration) by setting

-CONCNTR, MAXP, numb (default to 0)

And/or change default number of iterations resulting convergence inspite of penetration:

-CONCNTR, CNIT, numb (default to 4)

e.g.

CONCNTR,MAXP,3

CONCNTR,CNIT,5

MAX. NUMBER OF VIOLATIONS POINT. FOR OVER PENETRA. 3

NUMBER OF ITERATIONS IN SPITE OF PENETRATION 5

The control parameters can be reset in a subsequent step.

WARNING: These controls are intended for experienced analysts and should be used with great care.

Axisymmetric contact



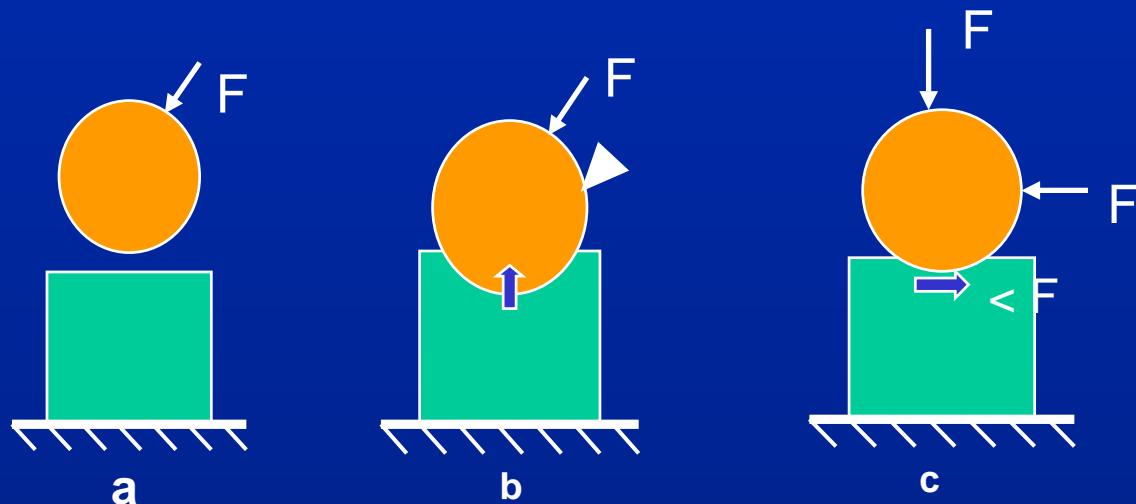
- Inaccurate contact stresses when using axisymmetric elements at the symmetry axis.
 - For axisymmetric elements the contact area is zero at the nodes lying on the symmetry axis when:
 - CONTA175
 - CONTA171,172 with KEYOP(4)=1,2
 - To avoid numerical singularity problems caused by a zero contact area, ANSYS provides a small area at that nodes. This may result in inaccurate local contact stresses for the node located on the symmetry axis.

Rigid Body Motion

ANSYS

Sometime, the parts in a contact model is not uniquely constrained, It means some parts can move without producing any elastic deformation- Rigid Body Motion. This rigid body motion is unfortunately not allowed in a static analysis. Usually, there are 3 types of rigid body motion

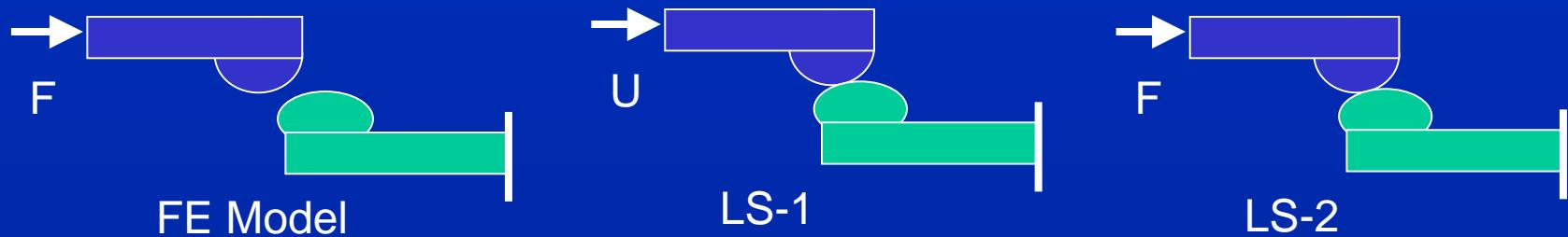
- The parts are not in contact
- The initial penetration is too high, this will leads to a very high contact force
- Friction force is too small comparing the external force



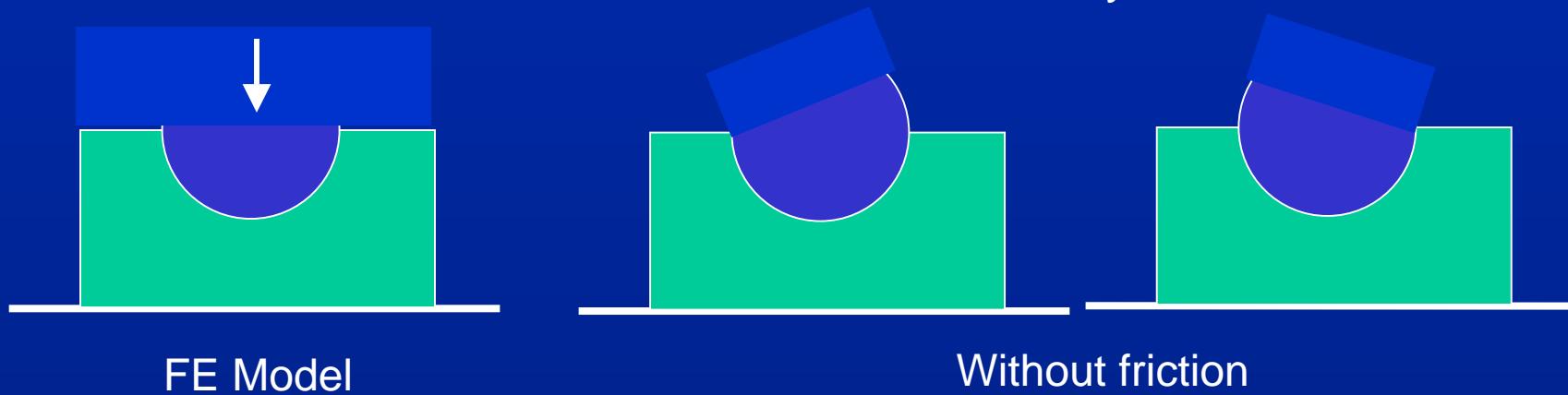
Rigid Body Motion Prevention

ANSYS

- Use reasonable boundary condition, e.g. use 2 load steps, load step 1 to move the body together and then apply the load.



- Add a small friction to avoid the numeric instability



Rigid Body Motion Prevention

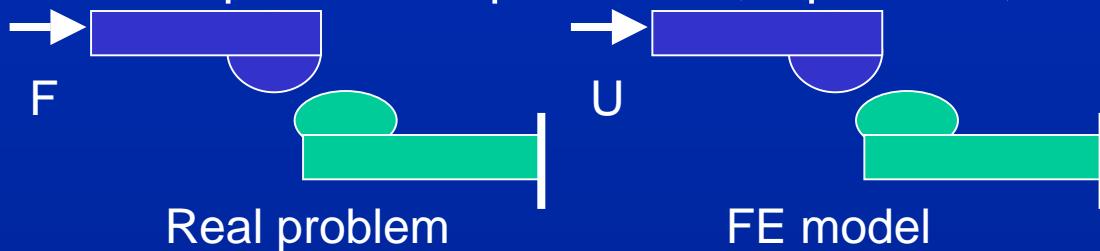
ANSYS

- Do a slow dynamic analysis to introduce the inertial force.



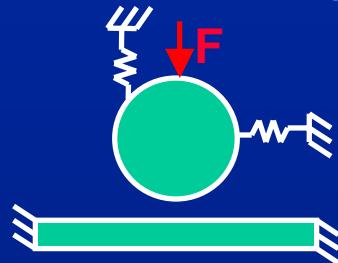
• You will need to add mass and damping in order to convert the solution from a static to a dynamic solution. This is known as a *slow dynamic* solution.

- Use specified displacement, if possible, instead of force.



This technique uses imposed displacements to move the body and as result you get the reaction force

- Use weak spring to avoid the numeric noise.

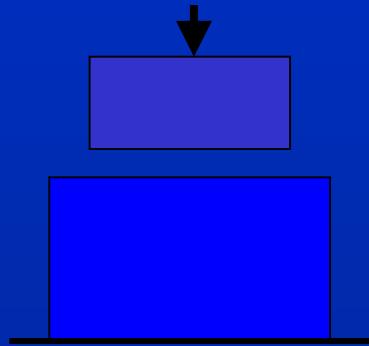


The spring stiffnesses should be negligible compared to the stiffness of the system. By connecting the springs to ground, the reactions at the grounded nodes can be compared to the total reaction forces to ensure that the springs have no effect on the solution. The open-spring contact option could also be used.

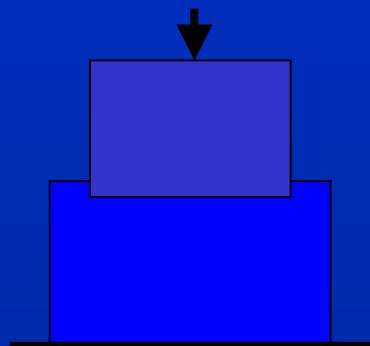
Rigid Body Motion Prevention

ANSYS

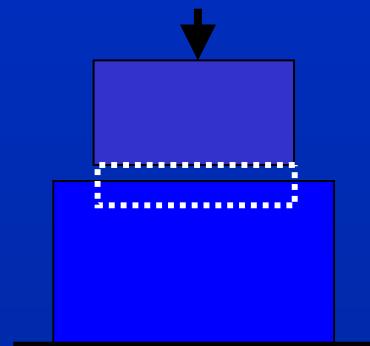
- Use artificial initial penetration to close the contact.



FE model with initial gap



FE model with
geometrical penetration



FE model with artificial
penetration

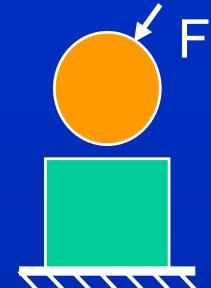
- Use no-separation Option KEYOPT(12)

Rigid Body Motion Prevention

ANSYS

TIP

Never try to solve a static problem with rigid body motion!



When possible, one should always use a very small geometric penetration to prevent the rigid body motion, at the same time, the geometric penetration establishes the initial contact region, which hold the structure together to distribute the load.

The initial penetration can be modeled via:

- a) Overlap the geometry slightly, be careful in plastic analysis, because even the small penetration can produce plasticity.
- b) Use CNOF to close the geometric gap, it can be very tedious, because you must know the gap size to adjust the CNOF.
- c) Use KEYOPT(5) to close the geometric gap automatically.
- d) Use Contact178, instead of surface contact, Contact178 can ignore the all the gaps

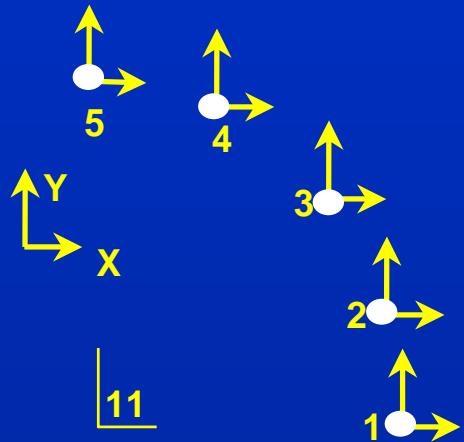
Initial Contact Condition Adjustment



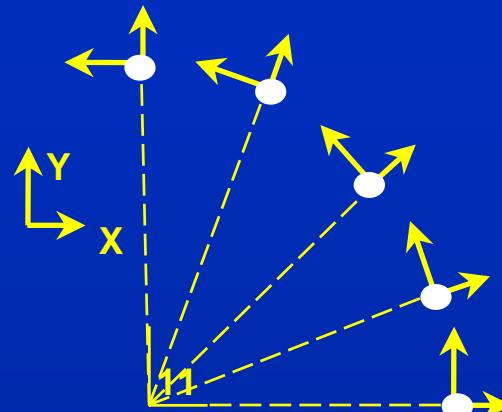
- User defined contact surface offset CNOF
- Automatic adjustment using KEYOPT(5)
- Adjustment band ICONT - making in contact
 - Always check default value
 - Turn off for other strategies to be effective
- KEYOPT(9) to adjust initial penetration
 - Include, exclude, ramp
- Initial allowable penetration range (PMIN,PMAX)
 - Auto move target along unconstrained DOF
- Combination of above techniques

Rotated nodal coordinate system

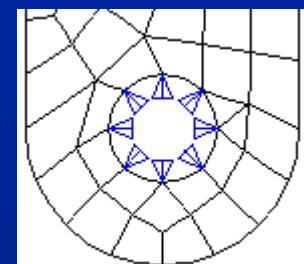
ANSYS



Default - parallel to global coordinate system

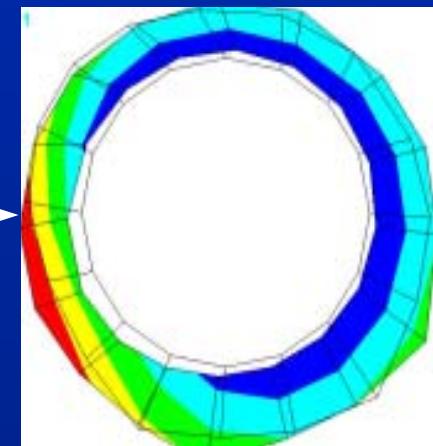
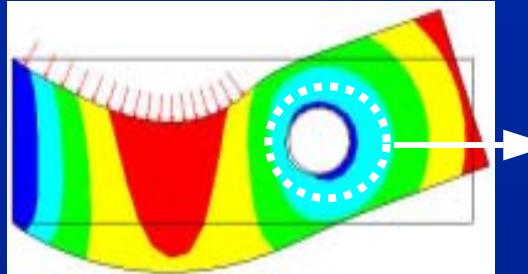
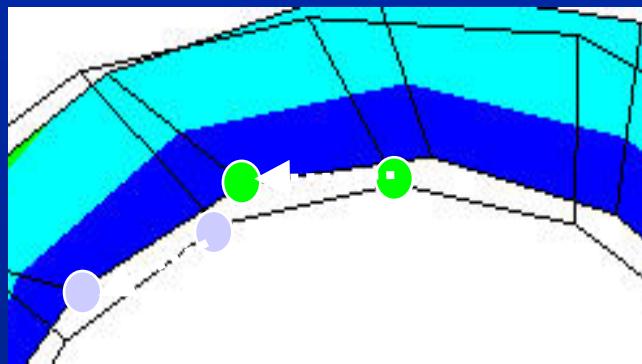
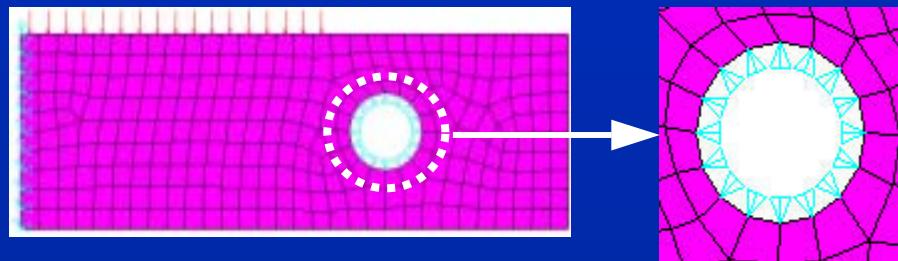


Rotated to local cylindrical coordinate system



Rotated nodal coordinate system

Caution: for large rotation analysis, the nodal coordinate system is not rotated to the deformed configuration. e.g. If node 1 is rotated into CSYS-1, the original nodal coordinate direction will not be changed.

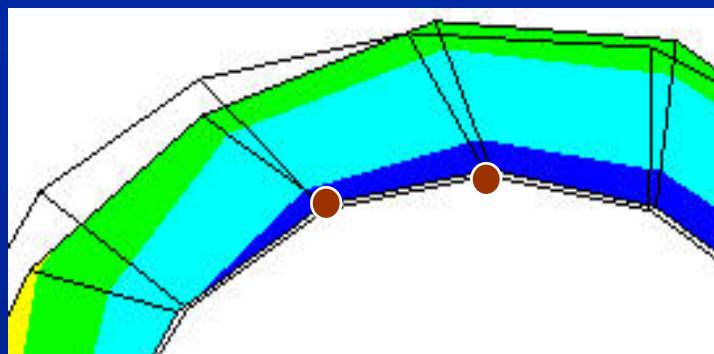
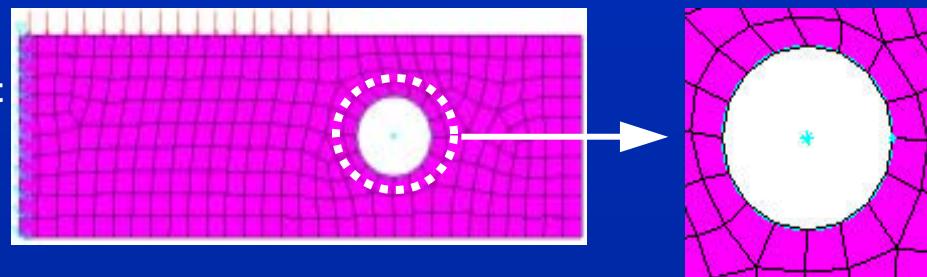


All nodes keep the original direction

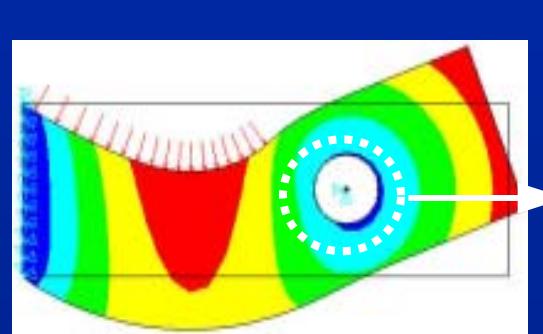
Rotated nodal coordinate system

This kind of problem can be nicely solved with contact element with Option:
Sliding only (KEY(12)=4).

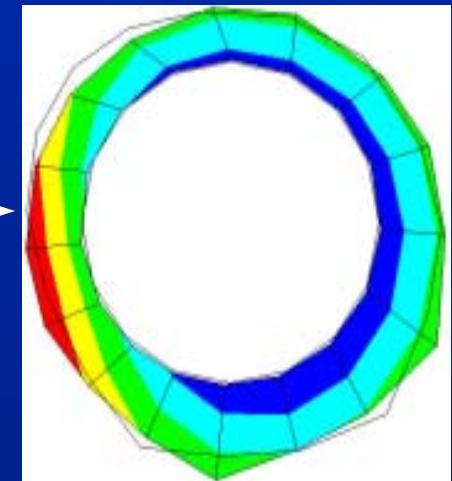
With Rigid-Target, all the nodes can only move in tangential direction.



Rigid-Target



The solution with contact elements



Conclusions



- A general framework is being implemented in ANSYS for finite element treatment of multi-body, multi-filed, large deformation frictional contact problems.
- We focus our study on robustness of algorithms, automation of analysis procedure, and diagnostic tools for troubleshooting.
- Today ANSYS is very capable of solving 3D self contact problems.
- Numerical results indicate the success of ANSYS contact technologies in solving very large scale engineering problems.

Conclusions



- The new surface based contact elements have many advantages over the node based contact elements
- Thanks to quadrature rules, 10-node tetrahedral and 20-node hexahedral solid elements can be used for contact analysis with a high degree of reliability.
- For geometrically nonlinear problems, a consistent approach was proposed which accounts for the change of geometry. It makes system equation converge quadratically.
- We have proposed a co-rotational system for the integration of friction laws which does not require to construct objective rates for the frictional tractions.
- Large sliding contact can generate unsymmetric stiffness terms and highly curved contact and target surfaces also produce large unsymmetric terms.

Conclusions



- For pure Lagrange multipliers method we have proposed
 - A new nodal quadrature rule in conjunction with mixed U-P formulation which well supports contact problems using higher order elements
 - Automatic contact tolerances to remedy chartering
 - Automatic resolution of overconstraint
- For Augmented Lagrange method we have provided
 - Better default setting for contact stiffnesses
 - Auto adaptive contact stiffness schemes
 - Contact elements superposition
 - More room to handle non-smooth surfaces, patch test, overconstraints
- Both methods have advantages and disadvantages which depend on the particular problem considered, could make either one look good or bad.