

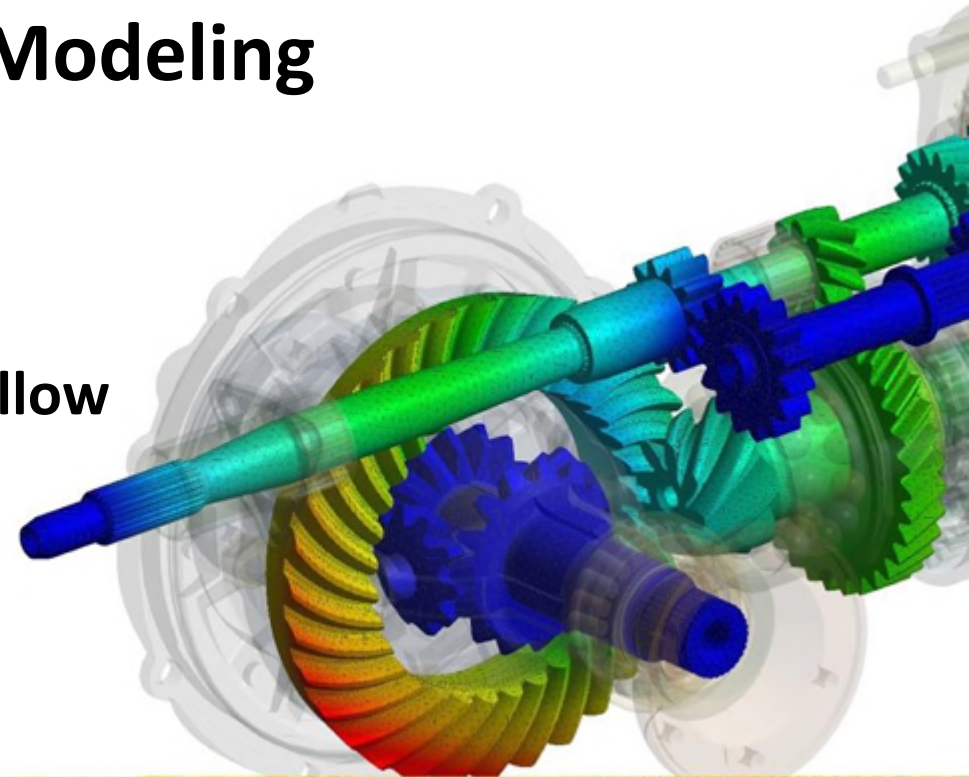


Best Practices for Contact Modeling using ANSYS

Yongyi Zhu, PhD

Research and Development Fellow

July, 2017

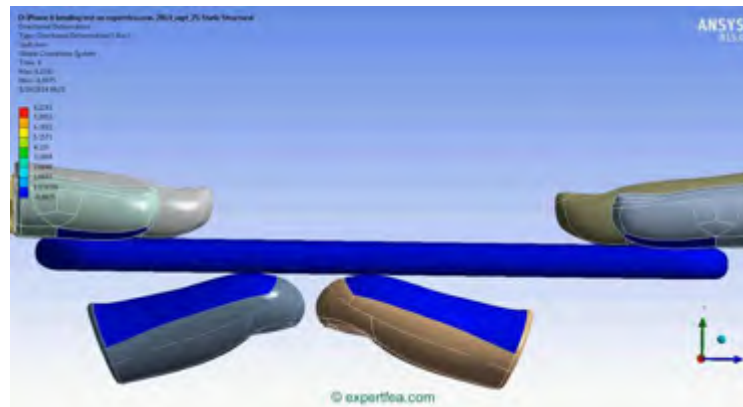


*This presentation contains ANSYS,
Inc. proprietary information.*

It is not to be distributed to others.

Why are these best practices important?

- **Contact is the most common source of nonlinearity and is often the most difficulty to solve!**
- **With typical structural problems, the presence of nonlinear contact can often be the biggest reason for increased solution times.**
- **Poorly defined contact may lead to unstable contact conditions. These conditions usually mean bad convergence and lost time.**
- **With ideal contact conditions, contact results converge much more quickly and the results tend to be smoother.**



What this presentation will cover

- **Section 1: Contact Model Setup and Verification:**
 - Contact Generation and Management Tips
- **Section 2: Getting Ready for the Solver:**
 - Mesh Quality & Mesh Sizing
 - Setting the Contact Formulation
 - Understanding the Effect of Contact Stiffness
 - Advantages of MPC Contact
 - Specifying Pinball Radius
 - Initial Interface Adjustment
- **Section 3: Dealing With Non Convergence**
 - Diagnostic Tools
 - Contact Results Tool
 - Overcoming Rigid Body Motions
 - Procedure for Overcoming Convergence Difficulties

Set the Right Units System

- **Always verify the model units.** Bad units can result in inaccurate solution and bad convergence due to tolerance, precision, and numerical round off issues.
- **Checkout output:**
 - *contact length is too small*
 - *elastic moduli or force/mass quantities is too big*

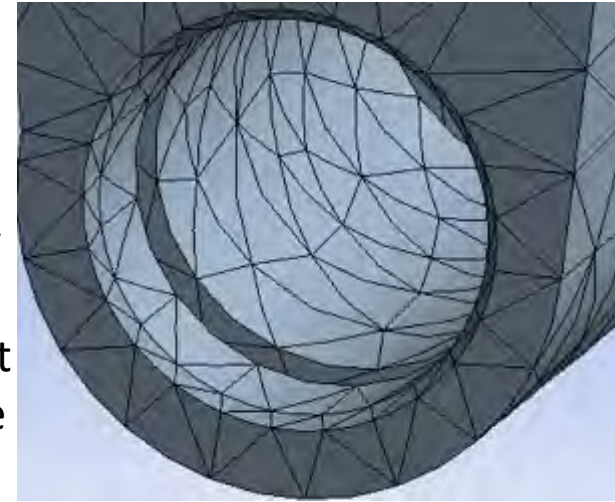


Tip: Select mm-N units for the most contact models.

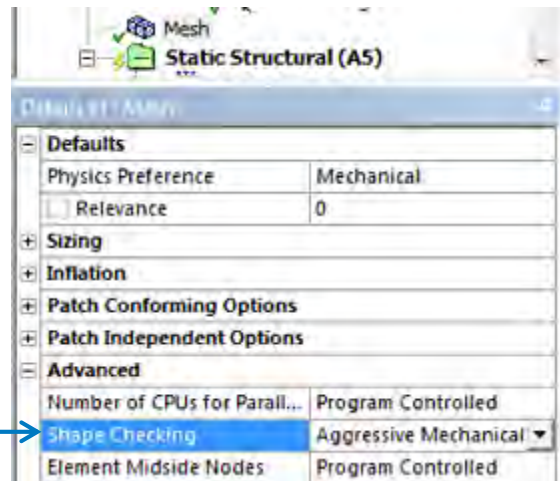
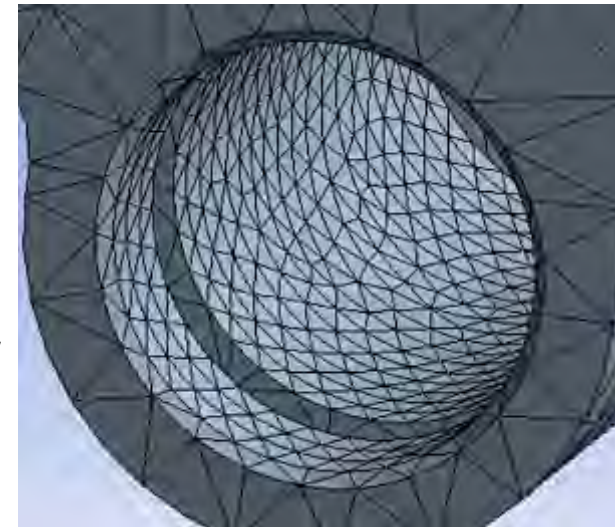
Mesh Quality

- Poor mesh quality in solid elements can cause convergence problems.
- A difficult contact problem may be diverging simply because of the mesh
- **Tip:** Use aggressive shape checking for nonlinear contact problems.

Poor Mesh Quality on Contact Surface

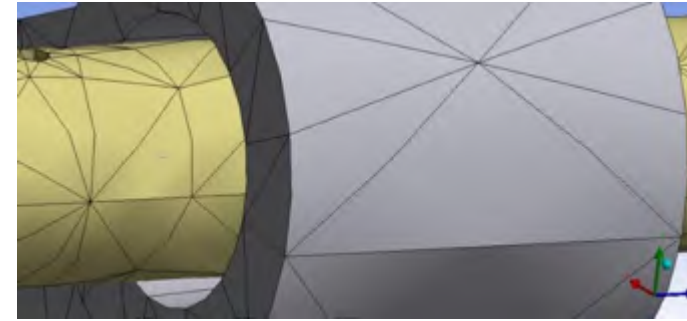


Better Mesh Quality

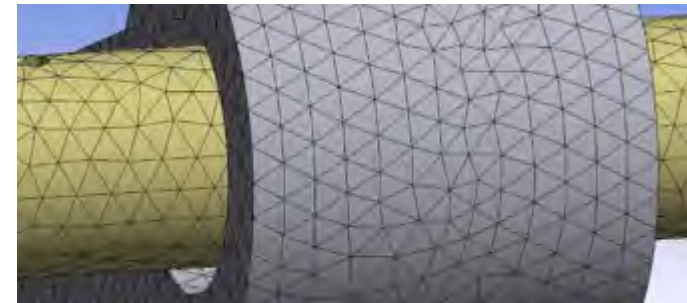


Mesh Sizing

- On curved surfaces, or surfaces which deform to a curve, having sufficient contact elements to closely follow the curvature is essential for smooth results.
- This is especially true for nonlinear contact
- Use similar element sizes for the source and target sides.
- **Tip:** Use Contact Sizing for sub-surface mesh control: Element size transitions very slowly!

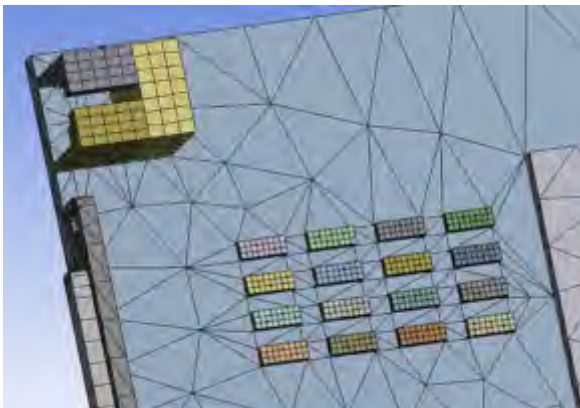


Too few elements

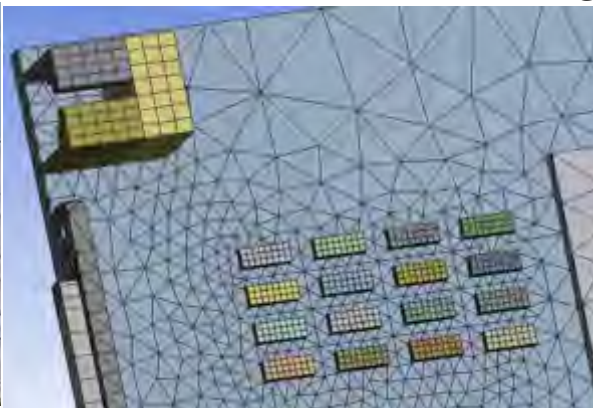


Better set elements with similar mesh density

without contact sizing



with contact sizing



Using Match Mesh Control

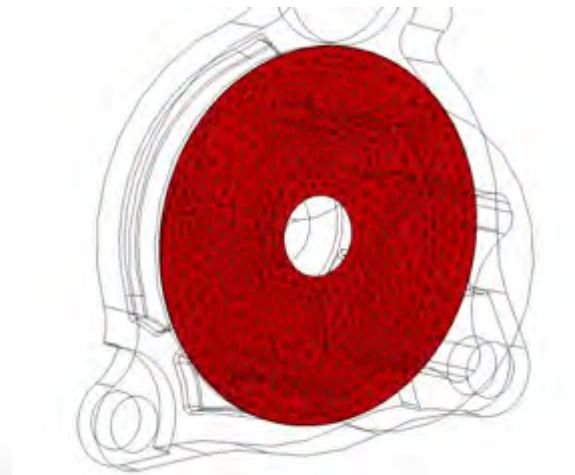
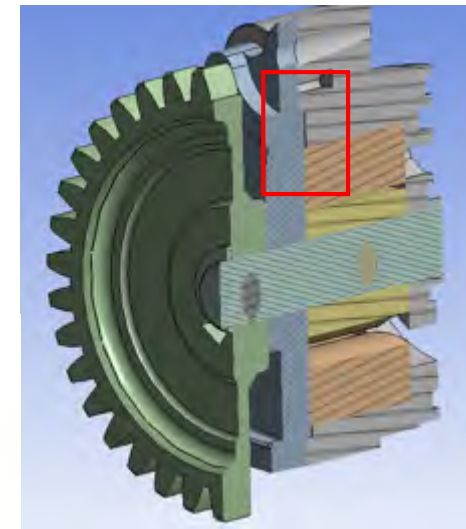
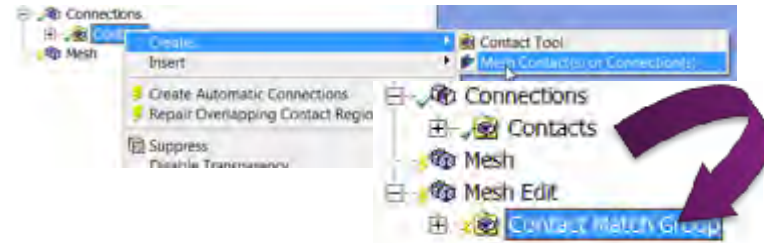
Mesh generated as separate parts

Use Contact Match controls to match mesh

Node merge can be added as second step to make mesh conformal

Tet Mesh Only

Tip:
Match mesh feature aids accurate definition of contact interface and improves convergence



Using the Worksheet

The worksheet view is a great way to review all of the contact settings



Rows can be sorted by clicking on column header

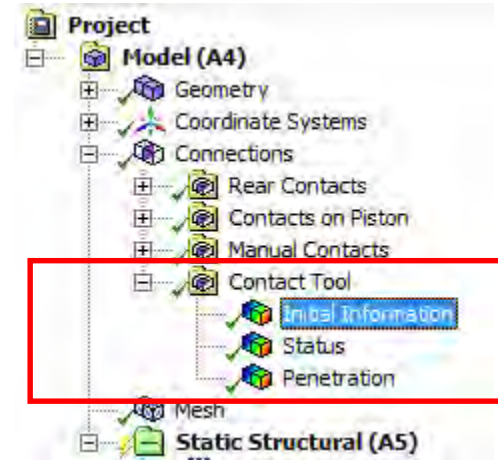
Column Visibility can be controlled via RMB

Name	Type	Friction Coefficient	Normal Stiffness	Update Stiffness	Formulation
Frictionless - Right Bracket To Input	Frictionless	N/A	Program Controlled	Program Controlled	Program C
Frictionless - Left Bracket To Input	Frictionless	N/A	Program Controlled	Program Controlled	Program C
Frictional - Right Bracket To Ujoint	Frictional	0.12	0.1	Program Controlled	Program C
Frictional - Left Bracket To Ujoint	Frictional	0.12	0.1	Program Controlled	Program C
Bonded - bolt2 To Input	Bonded	N/A	N/A	N/A	MPC
Bonded - bolt2 To Left Bracket	Bonded	N/A	Program Controlled	Program Controlled	Program C
Bonded - Left Bracket To bolt4	Bonded	N/A	Program Controlled	Program Controlled	Program C
Bonded - bolt1 To Input	Bonded	N/A	N/A	N/A	MPC
Bonded - bolt1 To Right Bracket	Bonded	N/A	Program Controlled	Program Controlled	Program C
Bonded - Right Bracket To bolt3	Bonded	N/A	Program Controlled	Program Controlled	Program C
Bonded - Ujoint To Input	Bonded	N/A	Program Controlled	Program Controlled	Program C
Bonded - Ujoint To Output	Bonded	N/A	Program Controlled	Program Controlled	Program C
Bonded - Input To bolt3	Bonded	N/A	N/A	N/A	MPC
Bonded - Input To bolt4	Bonded	N/A	N/A	N/A	MPC

- Type
- Scope
- Scope Mode
- Friction Coefficient
- Trim Contact Behavior
- Normal Stiffness
- Update Stiffness
- Formulation
- Detection Method
- Penetration Tolerance
- Elastic Slip Tolerance
- Trim Tolerance
- Interface Treatment
- Offset
- Damping Factor
- Pinball Region
- Time Step Controls
- Contact Geometry Correction
- Target Geometry Correction
- Status
- Reset Columns
- Reset Column Widths
- Edit Column Width

Using the Initial Contact Tool

- Use the Initial Contact Tool to quickly learn about contact status *before* solving.
- Inserted under Connection Folder
- Information reported which pairs are open/closed, how much initial penetration, inactive pairs, etc.
- Color coding to help user identify possible issues
- Contour results such as status and Penetration can be calculated



Name	Contact Side	Type	Status	Number Contacting	Penetration (m)	Gap (m)	Geometric Penetration (m)	Geometric Gap (m)	Resulting Pinball (m)	Real Constant
Bonded - V_MASTER_ROD[43] To V_MASTER_ROD_BEARING[44]	Contact	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	11.
Bonded - V_MASTER_ROD[43] To V_MASTER_ROD_BEARING[44]	Target	Bonded	Closed	224.	9.4299e-010	0.	0.	5.0152e-004	2.378e-004	12.
Frictionless - V_MASTER_ROD[43] To V_LINK_PIN[47]	Contact	Frictionless	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	13.
Frictionless - V_MASTER_ROD[43] To V_LINK_PIN[47]	Target	Frictionless	Near Open	0.	0.	4.9488e-004	0.	4.9488e-004	1.5135e-003	14.
Bonded - V_ARTICULATED_ROD[46,43] To V_ROD_BUSH_LOWER[48,44]	Contact	Bonded	Far Open	0.	0.	0.	0.	0.	3.1702e-004	21.
Bonded - V_ARTICULATED_ROD[48,43] To V_ROD_BUSH_LOWER[48,44]	Target	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	22.
Bonded - V_LINK_PIN[47] To V_ARTICULATED_ROD[46,43]	Contact	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	31.
Bonded - V_LINK_PIN[47] To V_ARTICULATED_ROD[46,43]	Target	Bonded	Far Open	0.	0.	0.	0.	0.	3.1702e-004	32.

Color Legend

Red	The contact status is open but the type of contact is meant to be closed. This applies to bonded and no separation contact types.
Yellow	The contact status is open. This may be acceptable.
Orange	The contact status is closed but has a large amount of gap or penetration. Check penetration and gap compared to pinball and depth.
Gray	Contact is inactive. This can occur for MPC and Normal Lagrange formulations. It can also occur for auto asymmetric behavior.

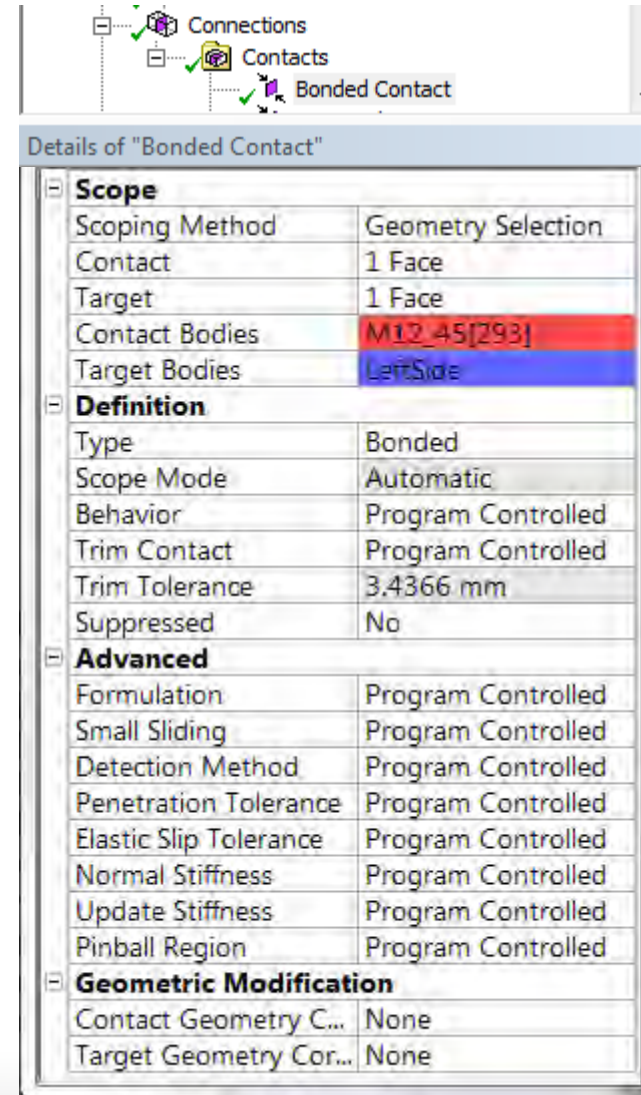
Section 2: Getting Ready for the Solver

Tip: Using Robust Defaults

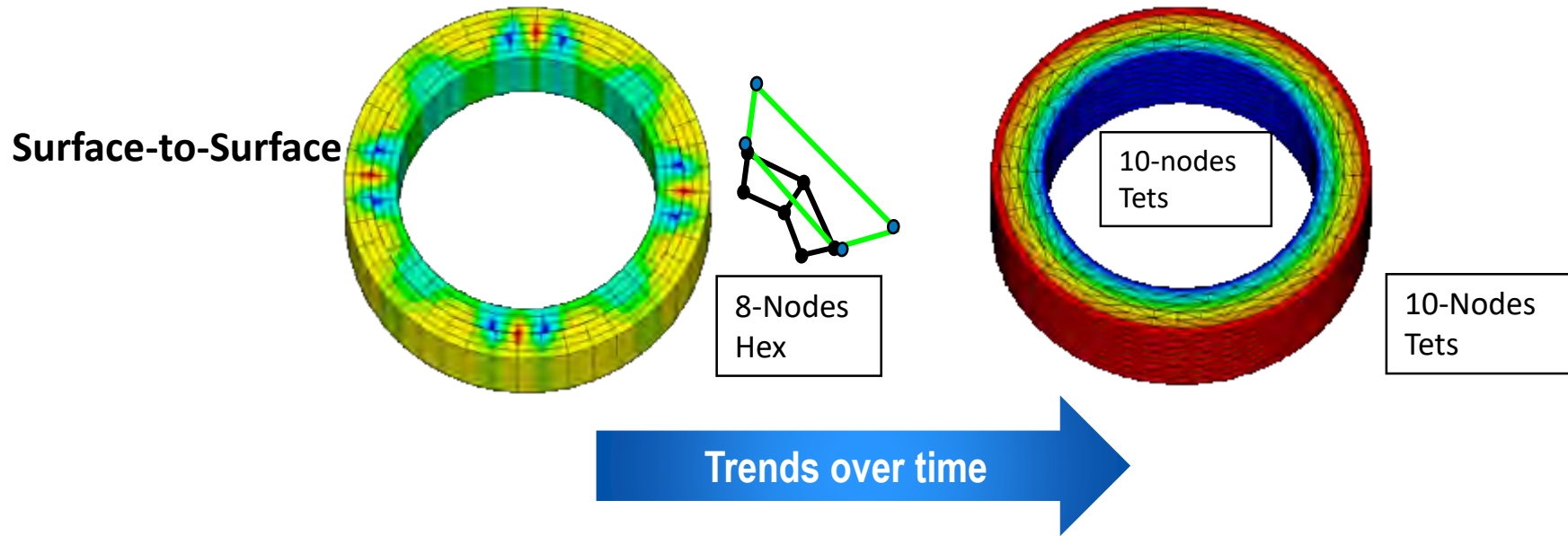
Robust defaults that solve the largest variety of contact situations is our goal

In general the “Program Controlled” settings in Mechanical match the MAPDL solver defaults.

Exceptions include Behavior(auto-asymmetric)



Contact Discretization: Linear Hex vs Higher order Tet



More efficient

Good for flat contact surfaces

Only for simple geometries whenever structure mesh is possible

Good for curve contact surfaces

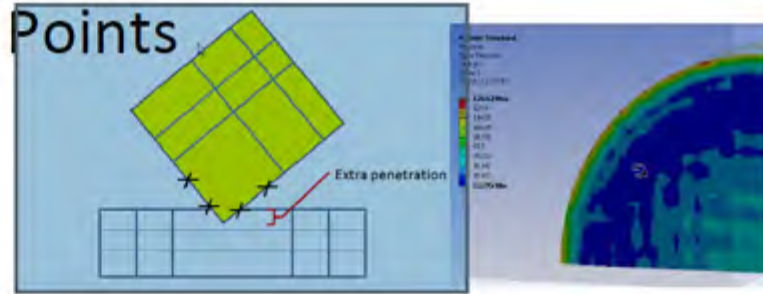
Good for Complex geometries with un-structure mesh

Smoother results

Contact Detection Methods

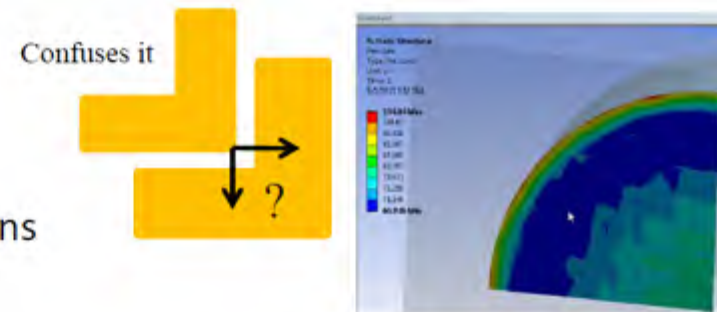
- Contact detection at Gauss Points

- Default
- Will have error at corners



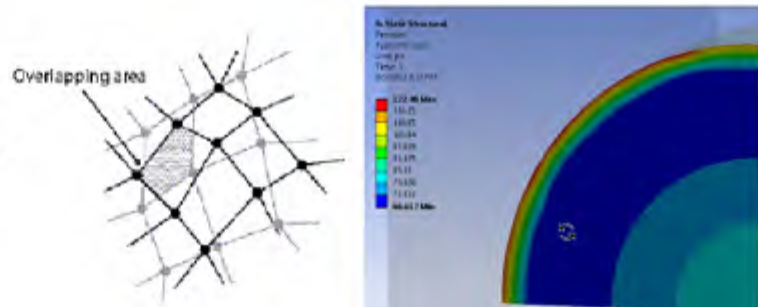
- Contact detection at nodes

- Smoother pressure plots
- Longer convergence times (30%?)
- Unstable for some surfaces combinations
 - Like two nested L-Shapes



- Nodal Projection from contact

- Latest and greatest

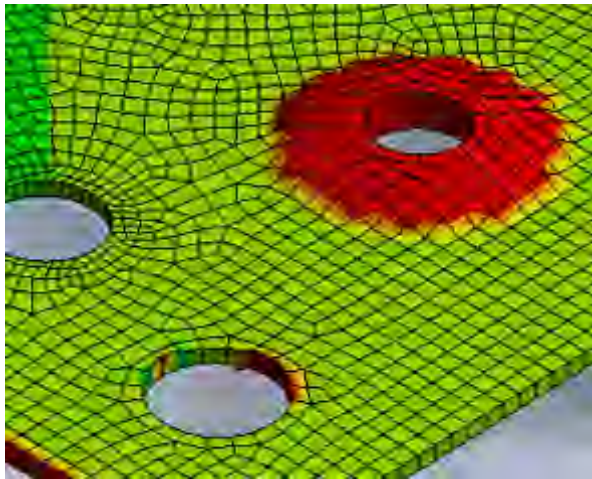


Contact Detection Methods: Notes

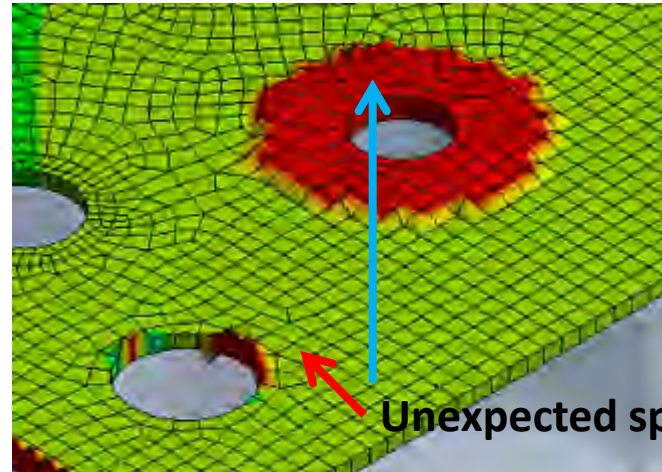
- Surface integration point method allow for additional points to detect penetration between surfaces - Default method for Penalty and Augmented Lagrange method.
 - However it is poor when contact occurs at corners or edges.
- Nodal based detection methods are default for MPC and Normal Lagrange method.
 - For contacts at corners (such as interference fit problems, threaded connector models), best results are obtained when either Nodal – Normal to Target (or) Normal for Contact.
- Nodal – Projected Normal From Contact
 - For true surface-surface contacts, it provides good results with minimal contact pressure spikes at nodes
 - Convergence behavior is also better if mesh is adequately discretized at contact surfaces.
 - For best performance, use similar mesh size for both contact and target surfaces.

Contact Detection Methods: Notes

- Nodal – Projected Normal From Contact
 - Always use Projection method for contact involving gasket layers so that stress and strain distribution near contacting edges is more smooth.
 - Always use Projection method for Acoustic FSI interaction.
 - Avoid using the projected contact in conjunction with MPC bonded contact when initial geometric gap/penetration exists. Close gap/penetration by issuing CNCH,MORPH or use the nodal detection method.
 - For 3D higher order elements, use Normal Lagrange in conjunction with projection based option for the best accuracy.



Using Projected contact



Using Gauss detection

Understanding Contact Formulation

- **Augmented Lagrange (Default):** Suitable for most problems.
- **Pure Penalty:**
 - Contact occurring only on Edge or Corner
- **MPC (Multi-Point Constraint):** Ideal for all bonded/no-separation contact when there is no over-constraint
- **Normal Lagrange:**
 - Highest accuracy
 - Contact with material nonlinearities
 - Between shells or thin layers
 - Interference fit
 - Large Sliding

Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	9.2434e-003 m
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Augmented Lagrange
Elastic Slip Tolerance	Pure Penalty
Normal Stiffness	MPC
	Normal Lagrange

Modeling Contacts: Normal Lagrange Method Notes

Penetration critical applications (accuracy).

Contact with predominant material nonlinearity.

Contact between shells or thin layers.

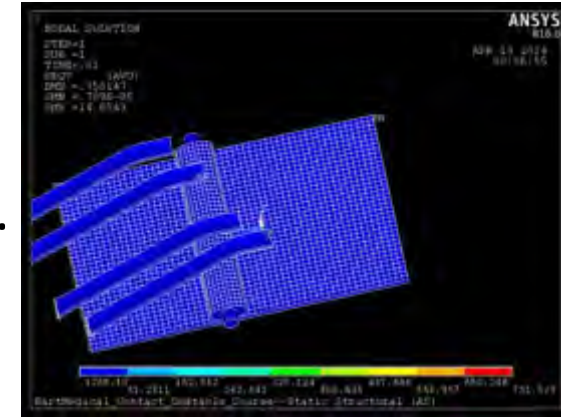
Large sliding problem. Pretension bolts

Small deformation due to small amount of loading.

Contact at corners, edges.

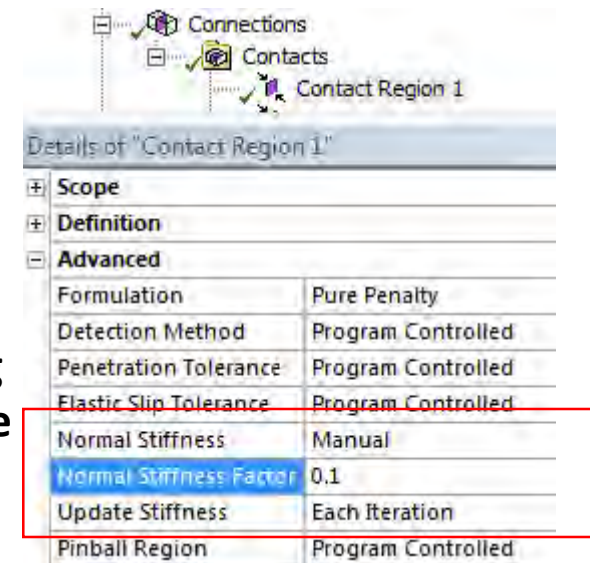
Suitable for solving threaded connectors, press fit joints, seals, etc. In which underlying stresses vary with contact stiffness.

Convergence is still not achieved after several attempts by adjusting contact stiffness.



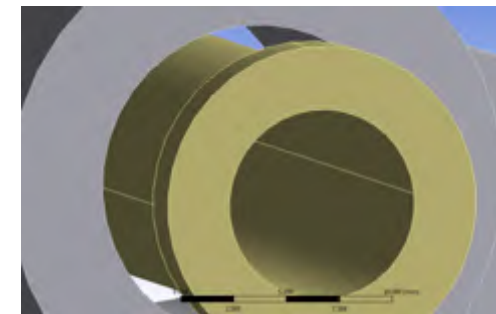
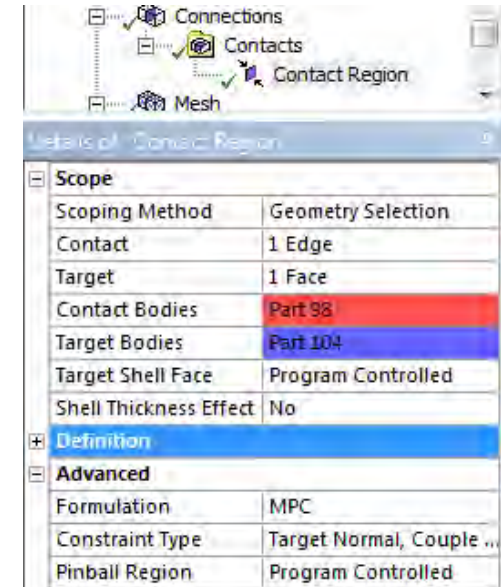
Contact Stiffness Factor: Tips

- For contacts with difficulty converging, lower the stiffness
- For pretension problems, use a stiffness factor greater than one, because penetration can strongly influence the pretension forces.
- Always set “Update Stiffness” to a frequency of “Each Iteration” (Default in WB Mechanical)
- Aggressive options: remedy chattering or preventing rigid body motion when initial contact conditions are not well posted.
- When there is difficulty converging due to high penetration, increase the stiffness.



Modeling Contacts: MPC Method Notes

- Use MPC contact for bonded and no-separation contact as much as possible unless overconstraints exist.
- Ideal for shell-solid, shell-shell, and beam-shell contacts
- Interface with small gap/penetration
 - It avoids spurious frequencies for modal analysis
- Gaps are frequently encountered in CAD models
 - Seam Weld preparation
 - Suppressed auxiliary parts (e.g. seals)
- Rather than modify the geometry to fill the gap, they can be accurately ignored when using MPC Contact or a fixed joint.
 - Joints/beam connection are recommended if convergence difficulties arise for large rotation problems.



Viewing the MPC Equations

Tip: Check overconstraints

Contact status

- Over Constrained
- Far
- Near
- Sliding
- Sticking

Outline

- Project
- Model (A4)
- Static Structural (A5)
- Solution (A6)

Details of "Solution Information"

Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Conne...
Draw Connections Attached To	All Nodes
Line Color	Connection T...
Visible on Results	No
Line Thickness	Single
Display Type	Lines

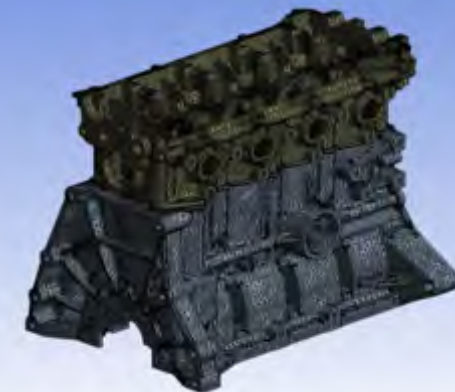
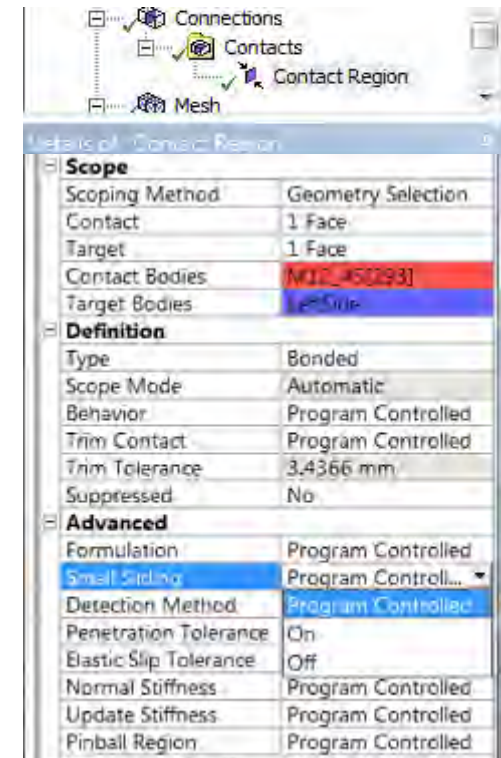
A: Static Structural
Solution Information
4/15/2014 4:29 PM

- Constraint Equation
- Beam
- Spring

After the solution is done MPC equations and other "FE Connections" can be graphically viewed

Small Sliding vs Large Sliding Assumption

- Finite-sliding option allows for arbitrary separation, sliding, and rotation of the contact interfaces. It is computationally expensive, but guarantees solution accuracy.
- Small-sliding option assumes relatively small sliding motion (<20% contact length) occurs on contact interface, but arbitrary rotations of contacting bodies is permitted. It improves solution robustness for models having a bad quality geometry or mesh and non-smooth contact interfaces.
- The small-sliding logic can cause nonphysical results if the relative sliding motion does not remain small. Use the finite-sliding option if you are not absolutely sure that the small-sliding logic is appropriate.
- Use the small-sliding option with great caution:
 - EKILL command applied to contact/target elements
 - Debonding (TB,CZM)
 - A general contact definition
 - Rezoning (REZONE) or mesh nonlinear adaptivity (NLADAPTIVE)



True Linear Contact

Concept:

- For bonded and no separation contact, if no other nonlinearities exist in the model (plasticity, large deformation, or unilateral contact), a linear solution (no equilibrium iteration) is good enough to obtain an accurate solution.

Advantages:

- Reaction forces are always balanced comparing with the existing “one iteration NL solution” (NEQIT,1,forcer).

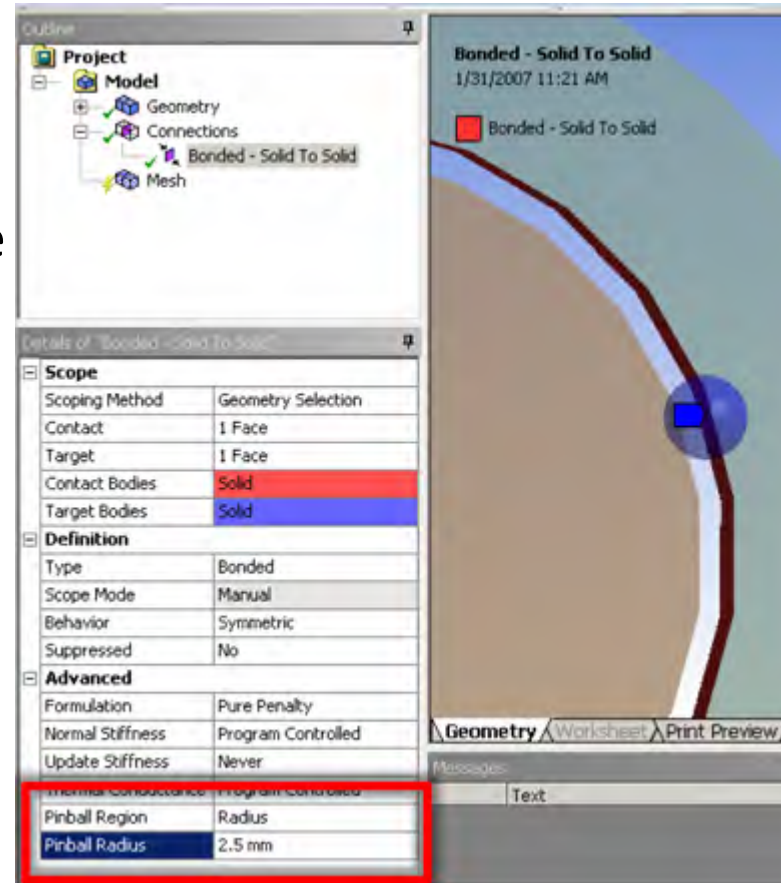
Applied force = 4N

Reaction forces	From contact elements	From underlying elements of contact	From underlying elements of target
Linear contact	4.002N (Reaction balance)	4.N	4.N
1 iteration with NL contact	6.4634e7 (Reactions don't balance)	4.N	4.N



Contact Pinball

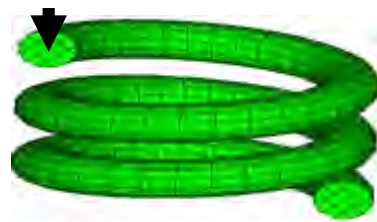
- ❑ ANSYS solver internally categorizes the state of contact as: Far field, Near field, Touching
- ❑ Category is based on if the distance between contact and target surface is lesser (or) greater than pinball radius
- ❑ For interference problems, ensure that the pinball radius is greater than the maximum interference
- ❑ In bonded contact and no-separation contacts, any region between the surfaces which touches or lies within pinball radius will be assumed to be in contact



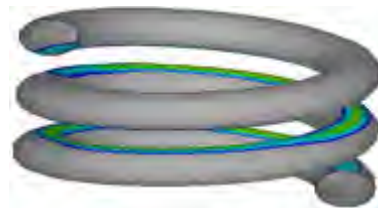
Contact Pinball - Notes

Pinball radius

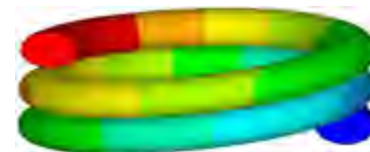
- This is one of the most important parameters to get the desired contact result
 - Effective PINB : measure the largest gap in your model via Initial Contact Tool, then specify PINB a little bit larger than the gap, e.g if the largest gap is X, use $PINB=X+X/10$.
- If you introduce a large pinball, you will have risks to introduce the spurious region.
- Use large pinball for load step which resolves interference fit. Use small pinball for other steps.



FE model



Contact status



Deformation

Contact Pinball - Notes

Usually, you should not use large PINB to run the analysis. The contact search time will increase.

Case 1:

User defined pinball region PINB 0.10000E-02

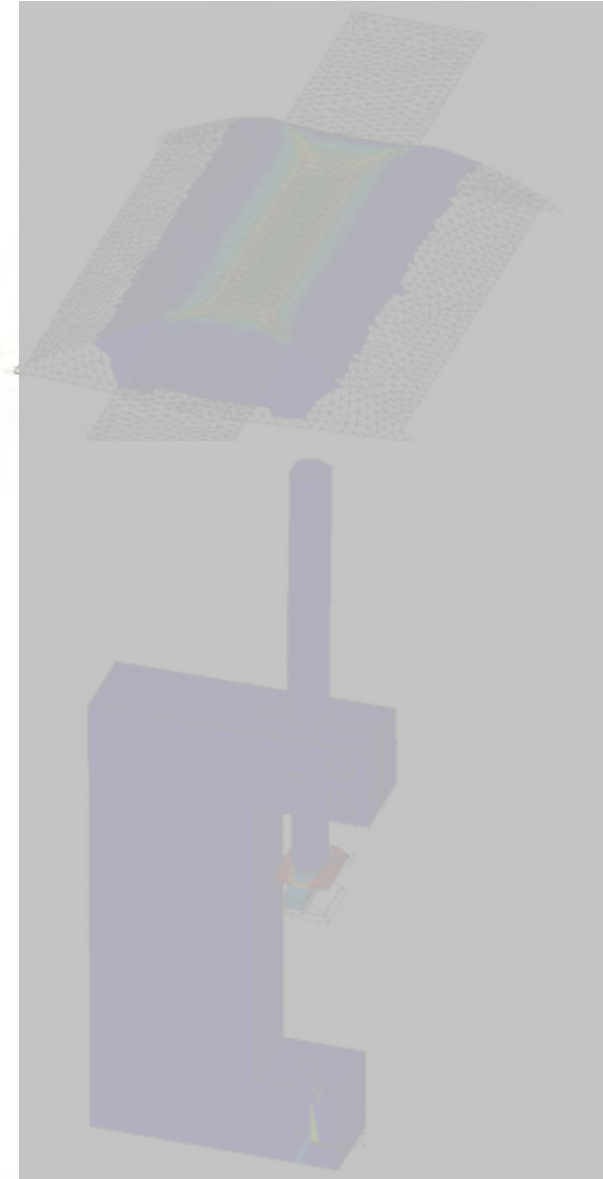
*** NOTE ***One of the contact searching regions contains least 348 target elements. You may reduce the pinball radius.

Case 2:

Default pinball region factor PINB 1.0000

The resulting pinball region 0.17553E-03

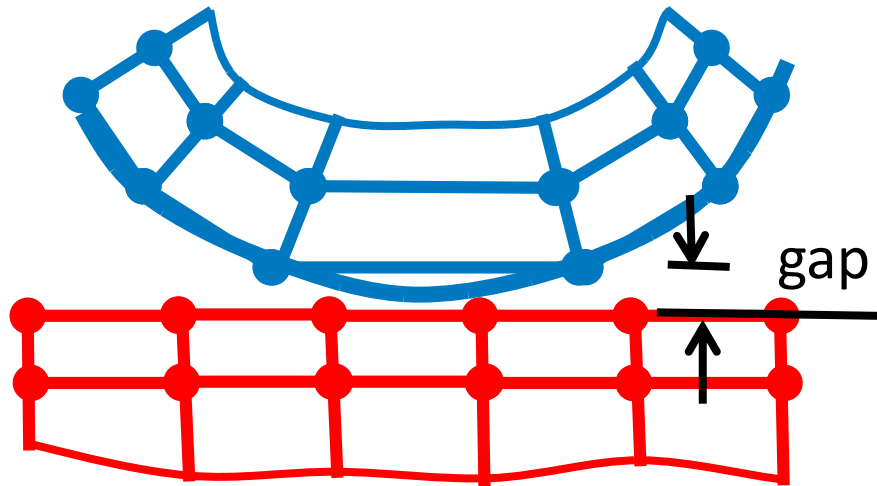
	CASE 1	CASE 2
CPU CONTACT SEARCH	2199.251	57.313
CONTACT ELEMENTS	399.591	236.996
OTHER ELEMENTS	328.731	227.731
EQUATION SOLVER	456.250	416.250
TOTAL SYSTEM	3383.140	938.290



CONFIDENTIAL

ANSYS

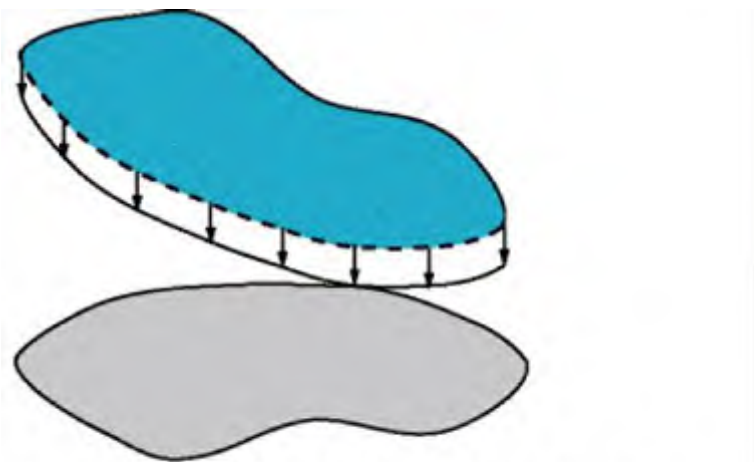
Interface Treatment: Adjust to touch



Initial mesh with undesired gaps and or penetration between contact surfaces results in difficulty in converge

Mesh discretization can create artificial gaps between surfaces even though the CAD surfaces touch each other

Adjust to touch option will remove the gap numerically and assume the surfaces touch each other

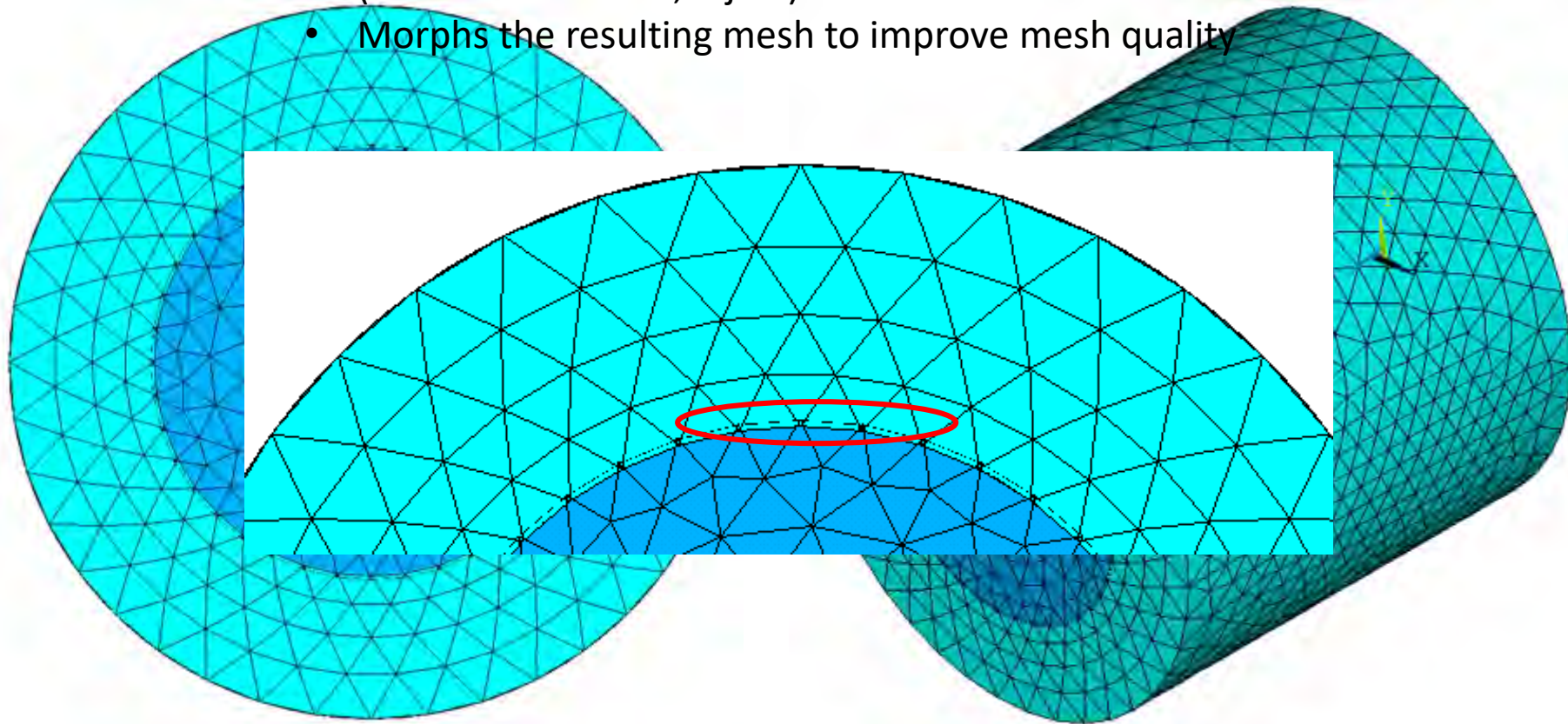


Contact pair after Adjust to Touch treatment.
Gap is closed automatically. Pair is "just touching".

Physically moving contact nodes towards target surface is not directly exposed in Mechanical, but can be manually defined via `CNCH,ADJU/MORPH` in a command object.

Mesh Morphing for Contact Adjustment

- Mesh morphing (Command: Cnch,morph) adjusts contact surfaces by stress-free movement of mesh
 - Moves the contact nodes to close gaps and remove penetration (similar to cncheck,adjust)
 - Morphs the resulting mesh to improve mesh quality



Cnch, morph creating initial interference

Modeling Interference Fit – Three Options

Contact surface offset (CNOF) as a function of time via tabular input

- Use tabular input to specify a table in which the magnitude of CNOF is ramped down from the possible maximum values of interference to zero over time.
- **For Arbitrary interface with varying contact normal**

Automatic interference fit method

- The program automatically ramps the initial penetration down to zero over time along the true contact normal direction.
- **For near flat interface**

User-defined contact surface normal

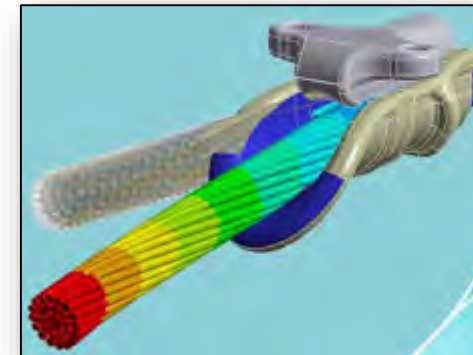
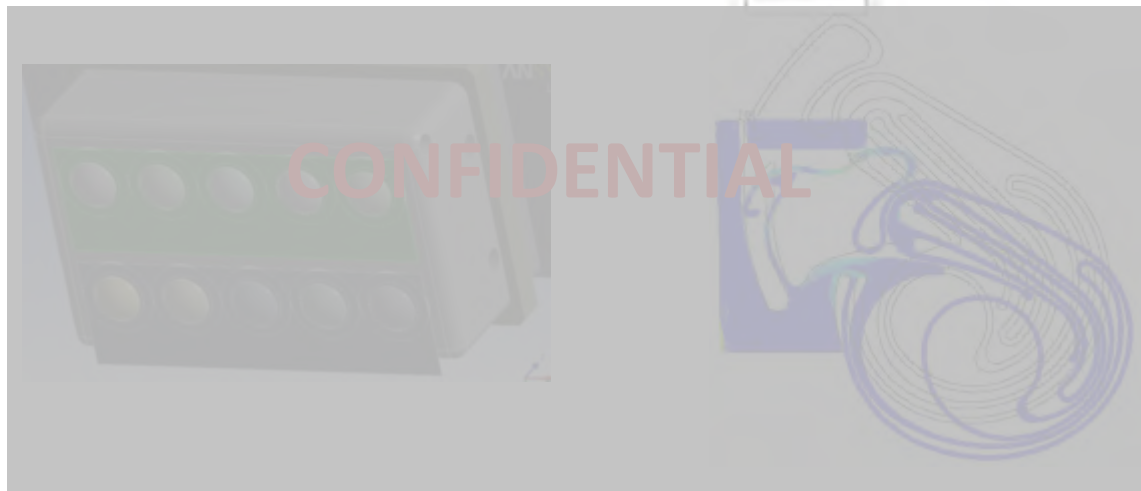
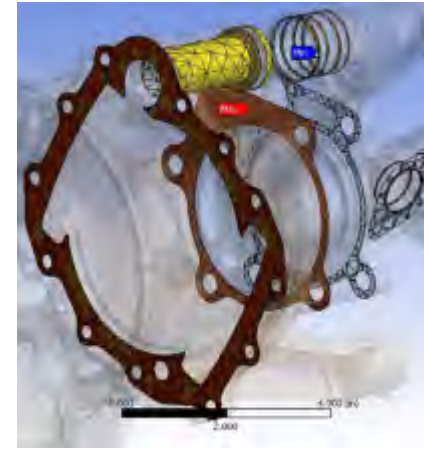
- The automatically ramps the initial penetration down to zero over time along the user-defined shift direction.
- **For curve interface**

Tip:

In some interference fit applications, the reactions calculated via contact element option may differ from those calculated via underlying elements. In such scenarios, the underlying element approach is more accurate. Try tightening the force tolerance (CNVTOL,F command) to improve the contact element reaction calculations.

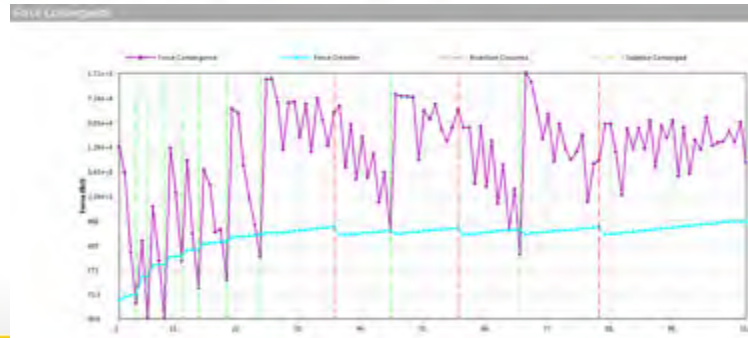
Contact Pair vs. General Contact

- The specific contact pair definition is usually more efficient and robust. It supports more options and special features. Take advantages of WB auto contact detection.
- The general contact automatically identifies all possible contacts. However, it is computationally expensive. It is recommended only for cases the pair definition cannot identify all possible interactions.
 - Complex assemblies, Large deformation/motion, Self Contact, Thin geometries



Section 3 : Dealing With Non Convergence

- First step is to determine the cause:
 1. Rigid body motion
 2. Force balance not obtained
 3. Material Instabilities
 4. Element formulation error
 5. Combination of items 1-4 above
- We will examine each in detail including:
 - identifying the problem
 - determining the cause
 - providing solutions



Reading the Solver Output

- Must read the output file!
 - Text describes convergence problems
 - Read it... it will begin to make sense!
 - Or send it to me and I'll translate
- Understand the Residuals Graph
 - This plots the NR residuals (error)
 - Abort non-converging runs
 - If it can't converge in 25-100 iterations it ain't gonna (usually)
 - If it can't converge at 1% (time =0.01) it ain't gonna (usually)

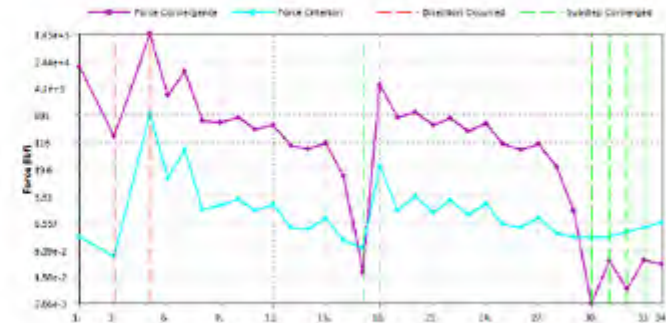
```
FORCE CONVERGENCE VALUE = 86.98 CRITERION= 7.365
EQUIL ITER 7 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.1830E-03
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.1830E-03
3D CONTACT ELEMENTS: 425 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE = 107.8 CRITERION= 8.576
EQUIL ITER 8 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.2239E-03
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.2239E-03
3D CONTACT ELEMENTS: 408 CONTACT POINTS HAVE TOO MUCH PENETRATION

RESTART INFORMATION

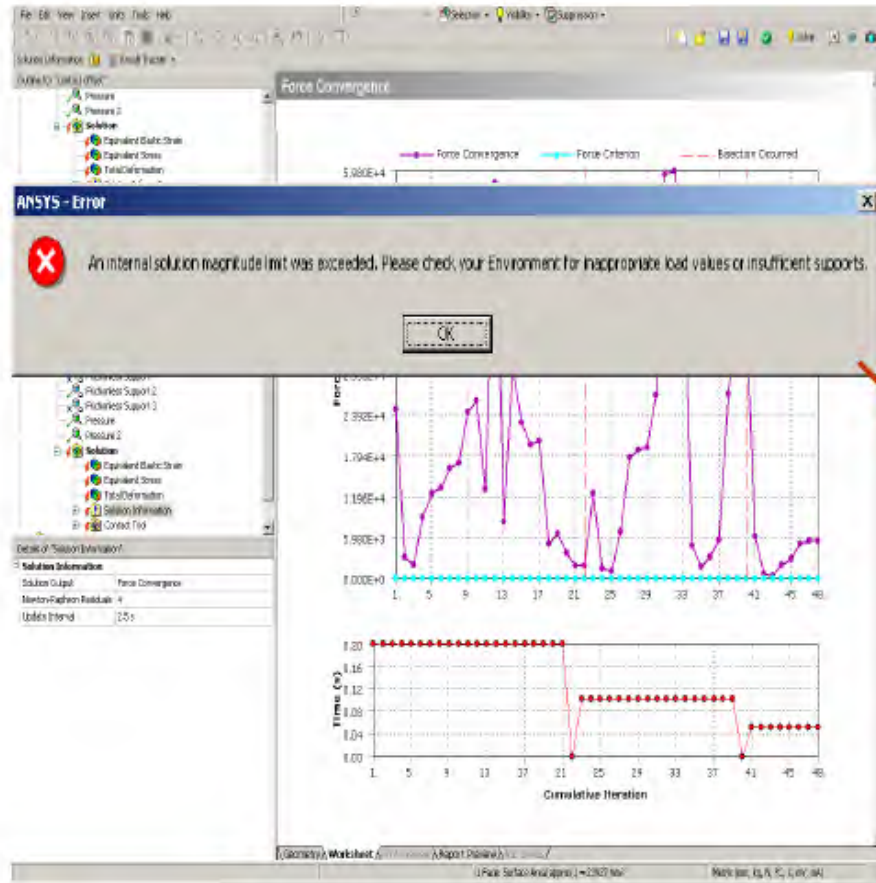
REASON FOR TERMINATION: DOF LIMIT EXCEEDED
RESTART BY RE-RUNNING THE ANALYSIS

ALL CURRENT ANSYS DATA WRITTEN TO FILE NAME= file.db
FOR POSSIBLE RESUME FROM THIS POINT

*** ERROR *** CP = 1254.288 TIME= 00:24:08
One or more elements have become highly distorted. Excessive
distortion of elements is usually a symptom indicating the need for
corrective action elsewhere. Try incrementing the load more slowly
(increase the number of substeps or decrease the time step size). You
```



Examples of Rigid Body Motion



```

*** WARNING ***                CP=   873.336   TIME= 13:07:32
There are 1 small equation solver pivot terms.
EQUIL ITER  7 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  -181.7
LINE SEARCH PARAMETER =  0.4042   SCALED MAX DOF INC =  -73.45
FORCE CONVERGENCE VALUE  = 5480.   CRITERION=  2.177
Writing NEWTON-RAPHSON residual forces to file: file.nr004
    
```

```

*** WARNING ***                CP=   897.090   TIME= 13:08:57
There are 2 small equation solver pivot terms.
    
```

```

*** ERROR ***                  CP=   897.230   TIME= 13:08:57
The value of UX at node 14305 is 2.393821741E+10.  It is greater than
the current limit of 1000000.  This generally indicates rigid body
motion as a result of an unconstrained model.  Verify that your model
is properly constrained.
    
```

```

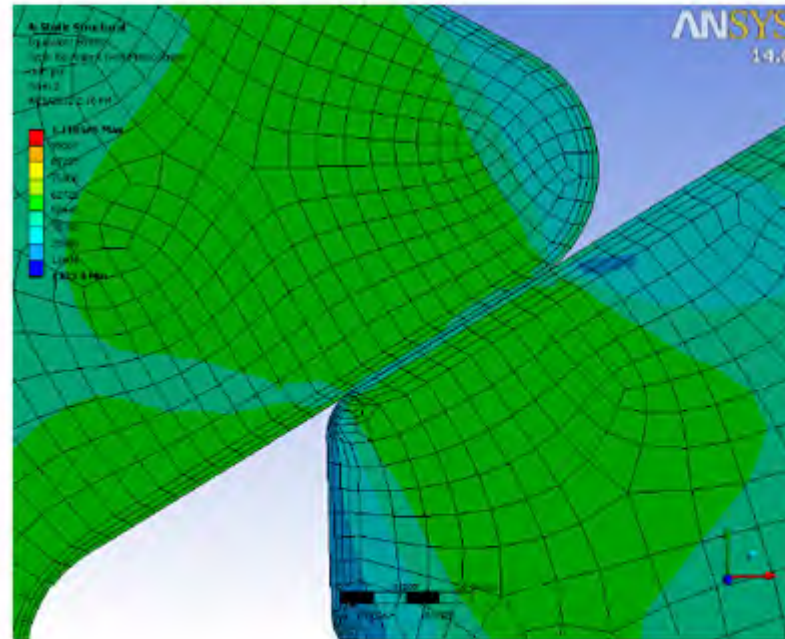
*** ERROR ***                  CP=   897.230   TIME= 13:08:58
*** MESSAGE CONTINUATION ---- DIAGNOSTIC INFORMATION ***
If one or more parts of the model are held together only by contact
verify that the contact surfaces are closed.  You can check contact
status in the SOLUTION module for the converged solutions using
CHECKCK.
    
```

```

*****
SUMMARY FOR CONTACT PAIR IDENTIFIED BY REAL CONSTANT SET          4
Max. Penetration of -2.734531805E-05 has been detected between contact
element 7173 and target element 7449.
For about 100 contact elements, there are 50 elements are in contact
    
```

Correct the Rigid Body Motion

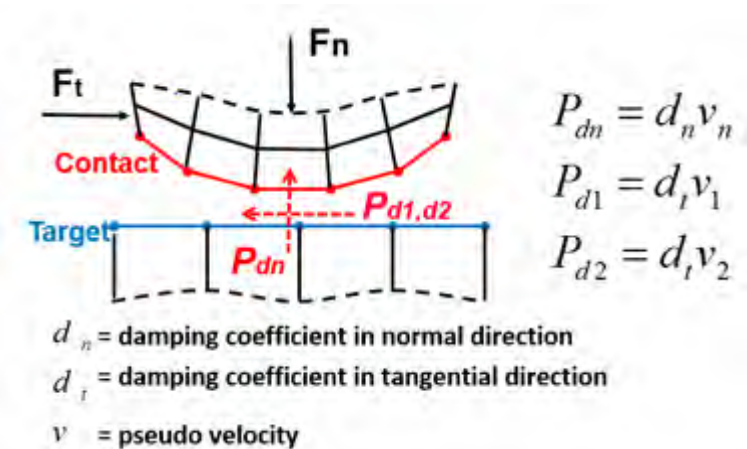
- Use bonded contact for debugging
 - If interfaces are all in compression, one might be able to leave as bonded
 - Modify to standard contact one pair at a time
- Adjust the parts to all start in contact
 - Can use adjust to touch ANSYS option, but be careful since the geometry will be changed
- Increase pinball region
 - Corrects interference fits
- Add friction
- Adding Contact Stabilization Damping



Contact Stabilization Damping

- Stabilization Damping Factor is applied in the contact normal direction and it is valid only for nonlinear contact (frictionless, rough, frictional contacts, and no-separate contact).
 - If this factor is 0 (default), the damping is activated under certain conditions mentioned and only in the first load step.
 - If its value is greater than 0, the damping is activated for all load steps.
 - Additional controls are available via KEYOPT(15) in a command object.
 - Tangential damping factor is not directly exposed in Mechanical, but can be manually defined via RMODIF in a command object.

Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Interface Treatment	Add Offset, No Ramping
<input type="checkbox"/> Offset	0. in
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	5.e-002

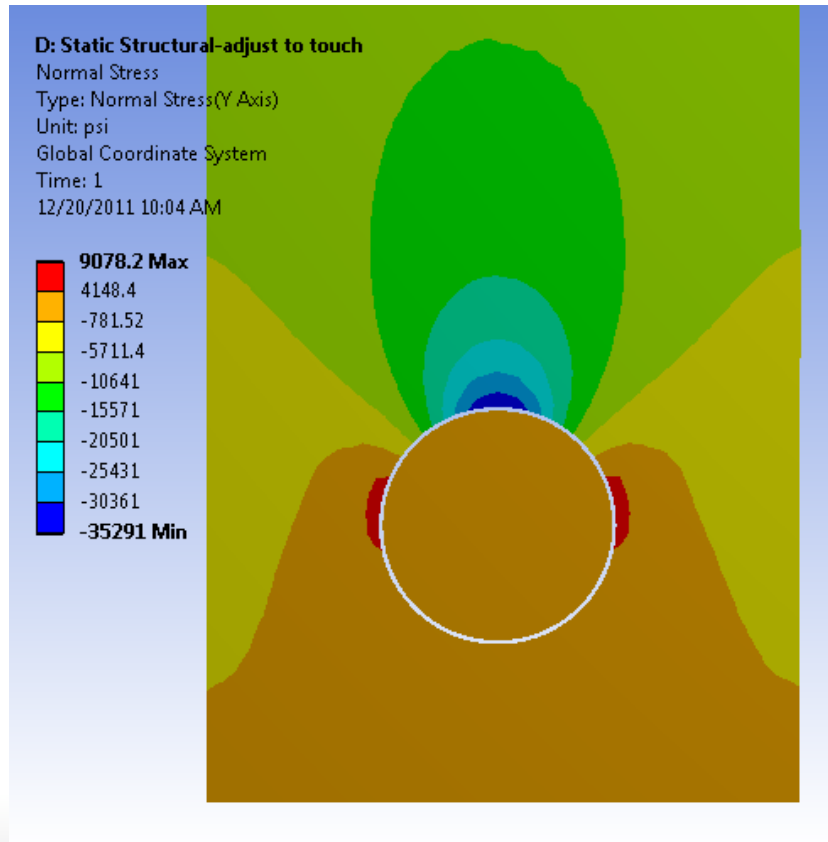


...Contact Stabilization Damping

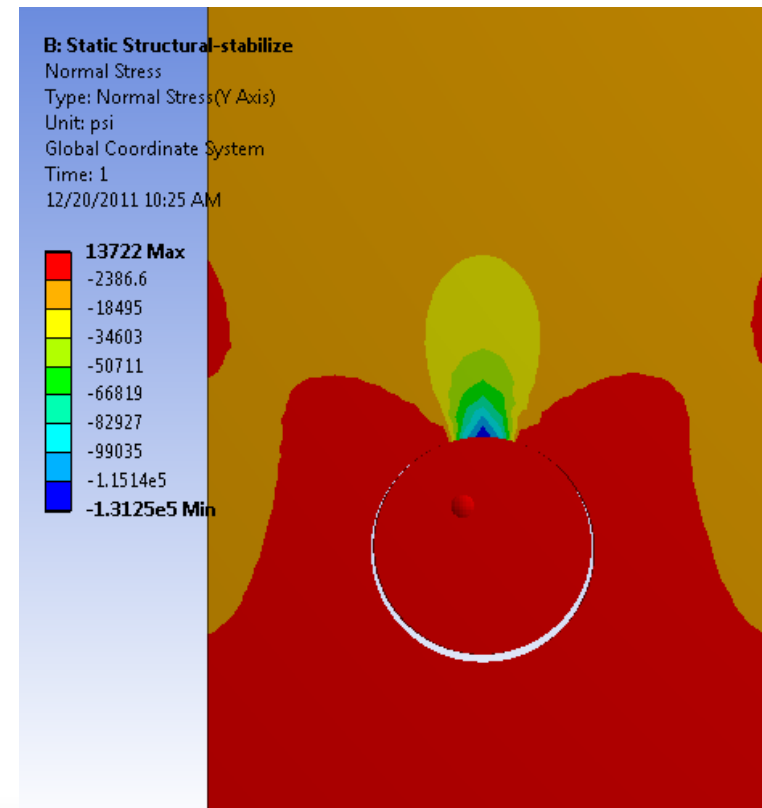
Example: Consider a fixed pin interfacing with a hole in plate with initial radial clearance and under a force based load

- Stabilization captures localized stress distribution more accurately than 'Adjust to Touch'

Conventional 'Adjust to Touch'



Contact Stabilization Damping



Debug strategies: Overconstraints

Overconstrained model

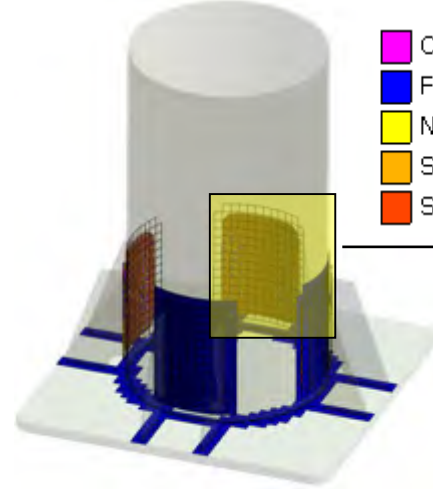
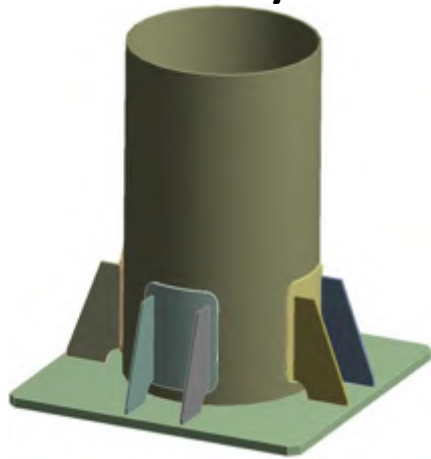
- Overconstraints are indicated by the presence of zero pivot warnings. It often results in very large residual force (orders of magnitude larger than a typically applied force) followed by very easy convergence.
- First check potential overconstraints via Contact Tool

Tips for manually removing overconstraints

- Remove overlapped pairs
- Merge pairs
- Flip contact and target surfaces
- Add multiple layers

Debug strategies: Overconstraints

The assembly is connected with MPC contacts

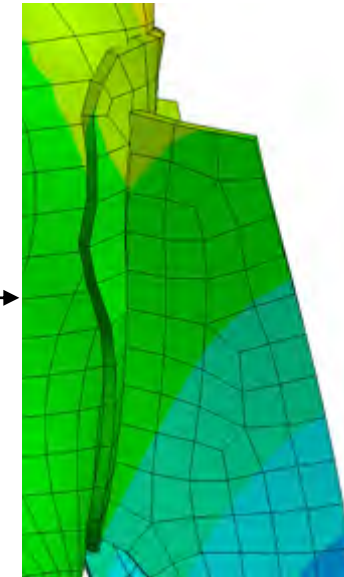


- Over Constrained
- Far
- Near
- Sliding
- Sticking

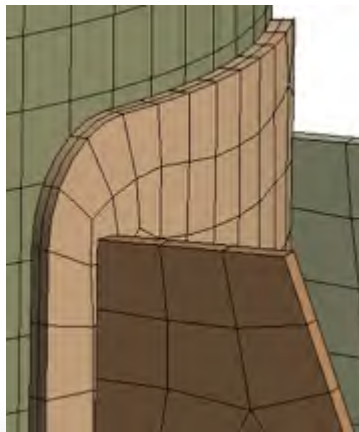


Contact status

In these regions Parts are not correctly connected



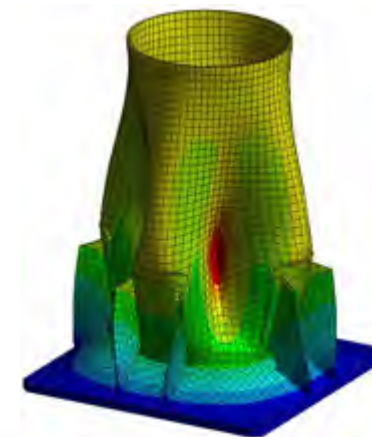
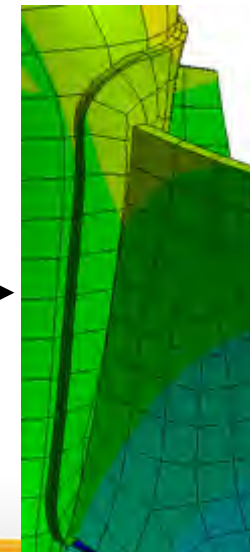
2 elements in sweep direction



- Over Constrained
- Far
- Near
- Sliding
- Sticking



Contact status



Solver Settings Decoded

Details of "Analysis Settings"

<input checked="" type="checkbox"/> Step Controls	
<input type="checkbox"/> Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
<input checked="" type="checkbox"/> Restart Controls	
<input checked="" type="checkbox"/> Nonlinear Controls	
<input checked="" type="checkbox"/> Output Controls	

Details of "Analysis Settings"

<input checked="" type="checkbox"/> Step Controls	
<input checked="" type="checkbox"/> Solver Controls	
<input checked="" type="checkbox"/> Restart Controls	
<input checked="" type="checkbox"/> Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Conv...	Program Controlled
Rotation Convergence...	Program Controlled
Line Search	Program Controlled
Stabilization	Off
<input checked="" type="checkbox"/> Output Controls	
<input checked="" type="checkbox"/> Analysis Data Management	

1) Weak Springs

- **Always turn it off**, unless it is really needed for prevent rigid body motion. Adding displacement constraints is the right way.
- When weak springs are used, carefully verify the reaction forces

2) Large Deflection

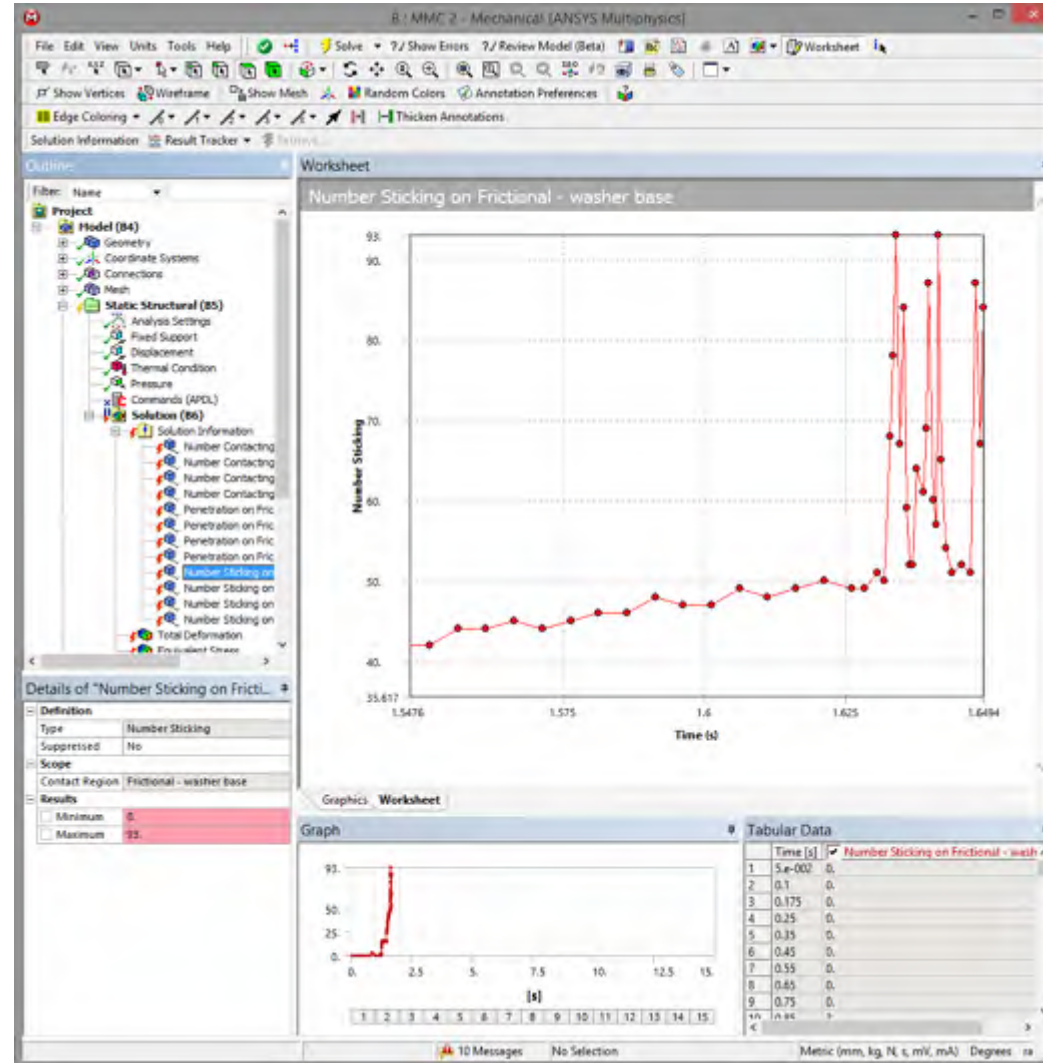
- **Always turn it on**, unless the model is truly for small strains, small deformations, small rotations, small sliding.

3) Stabilization

- Required for unstable problems
- Perform stringent checks on the results when stabilization is used

Diagnostic Tool: Contact Result Tracker

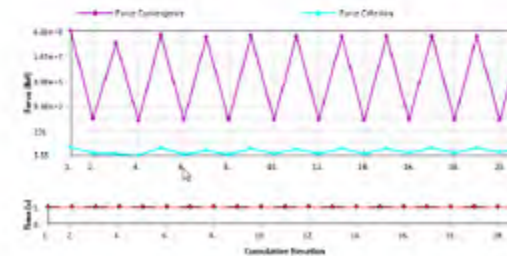
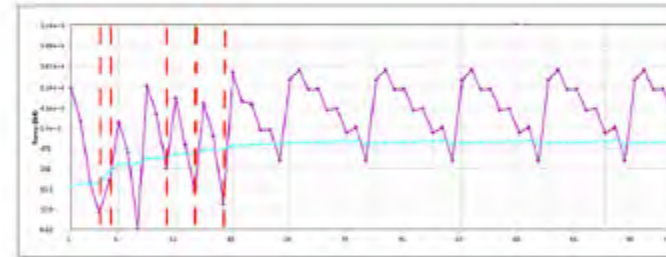
- Provides contact information during solution.
- The trends observed can help diagnose problems.
- For instance, a decreasing number of contact points indicate a loss of contact



Contact Results Tool: Chattering

- Contact Status Keeps Changing
 - Chattering (cycling of contact status in loop)
 - Often visible as NR residual pattern
- Try this
 - Reduce FKN (if penetration allows it)
 - Reduce time step size
 - Switch to Symmetric contact (or flip contact/target surface)
 - Refine the Mesh
 - Contact Stabilization
 - Change anything (numerical instability hole)

*** NOTE *** CP= 0.000 TIME= 00:00:00
6 contact points have abrupt change in contact status.



Overcome the Force Unbalance

- Reduce Normal Stiffness Factor

Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Interface Treatment	Add Offset, No Ramping
<input type="checkbox"/> Offset	0. in
Normal Stiffness	Manual
Normal Stiffness Factor	0.1
Update Stiffness	Each Iteration



- Increase the number of Substeps
 - General Rule – more nonlinearities – use more substeps

Define By	Substeps
Initial Substeps	100.
Minimum Substeps	20.
Maximum Substeps	5000.

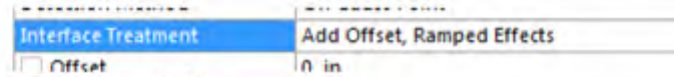
Solver Controls

Overcome the Force Unbalance

- Change contact stiffness update
 - Recommend using iteration based adjustments

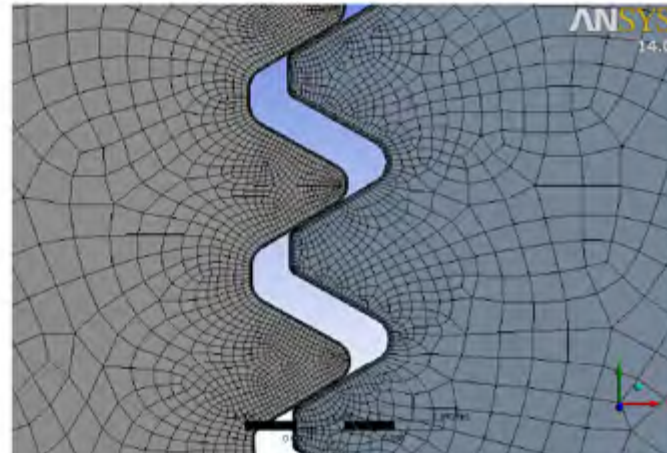
Normal Stiffness	Manual
Normal Stiffness Factor	1.e-002
Update Stiffness	Each Iteration

- Ramp on the interference fit



- Use Higher Order Elements
 - Especially with curved surface contact

- Refine the mesh
 - The more points in contact the better the convergence



Procedure for Overcoming Convergence Difficulties

- Identify the problematic contact region(s) using:
 - Contact tracking
 - Contact results (e.g. status, penetration)
 - Force convergence plots
 - NR residuals
 - Outputs
- Once identified, possible remedies:
 - Check Mesh Quality.
 - Make sure model units are on an appropriate scale
 - Adjust the Contact Stiffness
 - Check for proper initial conditions/pinball.
 - Change to nodal detection if the problem is at a corner.
 - Reduce the time step size before and during the onset of the divergence.
 - If large friction coefficient is defined(>.20) consider using unsymmetric solver
 - Use small sliding in conjunction with Lagrange multiplier method.
 - Last resort: Add stabilization(contact or global)

Debug strategies: Check non contact-related issues

- Unrealistic physical model
- Unreasonable loading and boundary conditions.
 - Insufficient constraints, missing rotation constraints
 - Overconstraints
 - Loading condition: Step vs. ramped (KBC)
- Element formulation: Hourglass and locking, U/P, joints
- Material constitutive Large plastic deformation, creep, incompressible or near incompressible
- Unreasonable or incorrect material properties and inconsistent units
- Local & global instabilities
- Follower loads, Link Elements

Debug strategies: Other Attempts

If above steps still don't provide enough information on what the problem is, there are other things that can be done:

- Turn on large deflection.
- Turn off weak springs.
- Use SPARE direct solver instead of PCG iterative solver.
- Turn off prediction.
- Solve problem using transient option with quasi static option.
- Use single process instead of multiple processes.

If none of above tricks works, call ANSYS experts for technical supports.