Advanced Techniques for Nonlinear Contact and Drop Shock Analysis
Shoubing Zhuang – Autodesk Inc.
Michael Smell – Autodesk Inc.

MA7405-P
Are you interested in learning how to use nonlinear contact to simulate cool contact models using Autodesk® Simulation Mechanical (formerly Algor®)? Would you like to know how to easily set up contact models through the complex contact settings and how to diagnose the convergence issues of nonlinear contact analysis? If so, then attend this class. This class will explore some of the challenges associated with nonlinear contact models. During this class, we will present a list of tips and tricks to help you breeze through your contact models, even if you are a beginner in nonlinear contact. These tips are summarized from vast experience solving customer issues. A variety of contact models will be used to demonstrate the key learning objectives.

Learning Objectives
At the end of this class, you will be able to:

- Recognize fundamental nonlinear contact terminology
- Define contact pairs in MES
- Identify and diagnose convergence challenges
- Create more efficient nonlinear contact models
- Perform a drop test analysis

About the Speakers
Shoubing is a senior research engineer for Autodesk® Simulation products at Autodesk in Pittsburgh. With 11 years of research experience on FEM, he has simulation expertise in computational contact mechanics, sheet forming simulation, and nonlinear finite element methods. Shoubing conducts research and development to improve the performance, quality, and reliability of surface contact analysis using mechanical event simulation in Autodesk Simulation software. He also provides FEA technical support to Autodesk Simulation application engineers and customers. Shoubing holds a PhD in mechanical engineering from The Ohio State University. This will be Shoubing’s first time to present at Autodesk University.
Shoubing.Zhuang@autodesk.com

Mike is a member of the Industry Management team at Autodesk and works on Market Development activities for the Autodesk Simulation product line. Mike has been with the organization for over 6 years with focus on simulation products and technologies. During this time he has led numerous classroom and customer-specific training sessions, overseen management of the virtual learning curriculum, and been involved in the daily sales and support of a broad range of simulation tools. As a member of the Industry Management team, Mike assists with customer adoption of our products, educates both our sales channel and customers on product offerings, and validates up and coming technologies for the company. Mike holds a Bachelor of Science degree in Mechanical Engineering, as well as a minor in
Introduction to Nonlinear Contact

Why do Contact Analysis?

Contact analyses can help
- Determine the contact areas
- Study deformation of contact parts
- Analyze the contact interaction
- Obtain contact forces

Contact analysis has been widely applied in many engineering industries, simulating mechanisms, rubber components, permanent deformation, impact, drop test, etc.

Capabilities…Analysis types that support Nonlinear Contact

Compared to a linear contact analysis, a nonlinear contact analysis has a much broader application, allowing the user to simulate contact conditions in models with large displacement, material non-linearity, large relative motion and dynamic instability.

Autodesk Simulation® has 4 nonlinear analysis types, of which the following three support nonlinear contact analysis
- MES with Nonlinear Material Model
- Static Stress with Nonlinear Material Models
- MES Riks Analysis

The capabilities of the three analysis types are summarized in the following table.
Contact Methods

There are 4 general methods of nonlinear contact available in Autodesk Simulation:

1. Frictionless Contact (default)
2. Frictional Contact
3. Slide/No Bounce Contact
4. Tied Contact

If a contact type other than frictionless is selected, i.e. frictional contact, slide/no bounce contact or tied contact, the user will have access to specific options in “Controls and Parameters for Contact Pair” dialog. Each of these options will give the user further control over the contact settings for each respective contact type.
Frictional Contact

The second most commonly used contact type is Frictional contact. When performing a contact analysis with friction, it is important to understand some of the details of how friction is applied to the model. In academics, friction problems are described using the Coulomb friction law. This approach is not well suited for FEA simulation as it can result in numerical instability making the problem more difficult to solve. Autodesk Simulation uses a Modified Coulomb friction law to describe the friction behavior in nonlinear contact. As you can see in the figures below, the Modified Coulomb friction law allows for a given amount of displacement based on the tangential contact stiffness as the friction force increases whereas the standard Coulomb friction law defines a near instantaneous change from static to dynamic friction coefficients as the model starts to displace.

Contact Types

Once the user has defined the most appropriate contact method as discussed above, there are some additional options for how the contact between the two bodies will be defined.

There are 4 general types:
   (1) Automatic (default)
   (2) Point to Surface
   (3) Surface to Surface
   (4) Point to Point

When the user chooses the “Automatic” option, the processor will choose between “Point to Surface” or “Surface to Surface”, based on the following selection criteria:
(1) Mesh sizes
(2) Contact surface shape
(3) Stiffness of contact parts

The “Point to Point” option best suits for the case in which the two contact surfaces have negligible relative sliding motion. The processor will automatically create elements between the nodes of the
contact surfaces and use these to solve the contact condition, see the figure below. As noted, this is most appropriate when there is negligible relative sliding motion between the parts as it is only monitoring the contact condition of each nodal pair. In the case of a larger relative sliding motion, if the nodes separate too far, they will go out of contact and penetration will occur. For comparison sake, the “Point to Surface” and “Surface to Surface” methods will monitor all of the nodes on one surface to the surface or the nodes of the other surface in the contact pair.

Defining Contact

There are two basic techniques for defining contact pairs in Autodesk Simulation. The first is to define contact pairs through a graphical selection process in FEA Editor using the following procedure:

1. Pick two contact surfaces
2. Right click in the canvas and select “Contact”
3. Select “Surface Contact” in the tree of “Contact”

The second method of defining contact pairs is to use the General Surface Contact dialog screen. This method does not use a graphical approach and requires that the user know the part and surface numbers that define each pair.
The procedure is as follows:

1. Right click in the canvas and select “General Surface-to-Surface Contact …” or select the Surface to Surface Contact button from the Setup tab of the ribbon.
2. Define the part and surface number of the first part in the contact pair.
3. Define the part and surface number of the second part in the contact pair.
4. Define the “Enabled” status of the contact pair as YES
5. Click “Add row” and repeat (2)~(4), if more contact pairs are needed
6. Click “OK” to close general surface contact dialog

Identifying and Diagnosing Convergence Challenges

Challenges of Nonlinear Contact Problems

The last section of this class discussed the available methods for defining nonlinear contact pairs. We will now switch our focus to solving the contact problem and how to identify and diagnose any problems we might encounter. Did you say problems with solving a contact model? “YES, that is what we said.” It is not unheard of to have troubles solving a nonlinear contact problem as the complexity of a problem increases exponentially as contact and freedom is added to the model. The most common factors that can lead to potential difficulty in the simulation are listed here:

1. Unknown contact state before analysis starts
2. Varying contact status
3. Additional complexity due to friction
4. Complex geometry
(5) Large relative motion
(6) Nonlinear material models

Possible warning messages to look for

Note that proper contact type can affect both accuracy and convergence of nonlinear contact models. If an improper contact type is defined, the processor will output some warning message in the log file:

(1) For a defined “Surface to Surface” contact pair:
** Warning: S-S contact is NOT necessary for contact pair, #!
P-S contact is recommended.

(2) For a defined “Point to Surface” contact pair:
** Warning: S-S contact is NOT proper for contact pair, #!
S-S contact is recommended.

(3) For a defined “Point to Surface” contact pair:
1st and 2nd parts in contact pair, #, should be switched.

If any of these warning messages are encountered, it is recommended to stop the model and make adjustments to the pairs that are identified based on what was discussed earlier in this document.

Diagnosing Convergence and Possible Issues

During an analysis, the step level, the number of iterations and convergence residual in the log file can be studied during the analysis to evaluate the performance of convergence. A step level of 1 will indicate ideal convergence based on the time step size that was defined in the Analysis Parameters. If convergence problems are encountered with a given time step size, the time step will be automatically reduced and the step level will increase until a solution is reached. While the analysis is running, the user can monitor these indicators in real time as this feedback is displayed in the Analysis screen. If the analysis is stopped, the user can also access the log file and summary file from the browser in the Report environment.

In the following figure, you will see headings that describe the information in each of the columns below. If you look closely, you will see the analysis was defined with a time step size of 0.04 seconds. Early in the analysis convergence is doing well as you can see the value in the L(Level) column is 1. As the model approaches 32% complete, it struggles to converge with a time step size of 0.04 and it gets reduced to 0.02 and the level value increases to 2.
When dealing with difficult convergence, it is also important to know where to find the parameters that are being used to define each of the contact pairs, especially when default settings are used during the setup. Accessing the summary file, the user can also check the values of contact stiffness and contact distance being used for each contact pair. In the coming sections we will be discussing how to interpret and adjust these values to improve convergence. See the following figure of the contact parameter section of the summary file.

<table>
<thead>
<tr>
<th>Time</th>
<th>Percent</th>
<th>DT</th>
<th>L</th>
<th>Iter.</th>
<th>Residual</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
<td>0.00</td>
<td>0.00</td>
<td>1</td>
<td>0</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>0.0400</td>
<td>4.00</td>
<td>0.04</td>
<td>1</td>
<td>1</td>
<td>1.9000E+00</td>
</tr>
<tr>
<td>0.0400</td>
<td>4.00</td>
<td>0.04</td>
<td>1</td>
<td>1</td>
<td>5.0000E+03</td>
</tr>
<tr>
<td>0.0400</td>
<td>4.00</td>
<td>0.04</td>
<td>1</td>
<td>2</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>1</td>
<td>5.0000E+00</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>1</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>2</td>
<td>6.0150E-02</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>2</td>
<td>6.1190E-02</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>3</td>
<td>3.2110E-02</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>4</td>
<td>7.9660E-02</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>5</td>
<td>2.5060E-02</td>
</tr>
<tr>
<td>0.1200</td>
<td>12.00</td>
<td>0.04</td>
<td>1</td>
<td>6</td>
<td>1.3900E-02</td>
</tr>
<tr>
<td>0.1600</td>
<td>16.00</td>
<td>0.04</td>
<td>1</td>
<td>7</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2000</td>
<td>20.00</td>
<td>0.04</td>
<td>1</td>
<td>8</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2000</td>
<td>20.00</td>
<td>0.04</td>
<td>1</td>
<td>9</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2400</td>
<td>24.00</td>
<td>0.04</td>
<td>1</td>
<td>10</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2400</td>
<td>24.00</td>
<td>0.04</td>
<td>1</td>
<td>11</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2400</td>
<td>24.00</td>
<td>0.04</td>
<td>1</td>
<td>12</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2400</td>
<td>24.00</td>
<td>0.04</td>
<td>1</td>
<td>13</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2800</td>
<td>28.00</td>
<td>0.04</td>
<td>1</td>
<td>14</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2800</td>
<td>28.00</td>
<td>0.04</td>
<td>1</td>
<td>15</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2800</td>
<td>28.00</td>
<td>0.04</td>
<td>1</td>
<td>16</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.2800</td>
<td>28.00</td>
<td>0.04</td>
<td>1</td>
<td>17</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>18</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>19</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>20</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>21</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>22</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>23</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.3200</td>
<td>32.00</td>
<td>0.04</td>
<td>1</td>
<td>24</td>
<td>1.0000E+00</td>
</tr>
</tbody>
</table>

When dealing with difficult convergence, it is also important to know where to find the parameters that are being used to define each of the contact pairs, especially when default settings are used during the setup. Accessing the summary file, the user can also check the values of contact stiffness and contact distance being used for each contact pair. In the coming sections we will be discussing how to interpret and adjust these values to improve convergence. See the following figure of the contact parameter section of the summary file.
Improving Performance of Nonlinear Contact

Now that we understand that “YES,” there can be difficulties encountered when trying to solve a nonlinear contact problem and what to look for when convergence issues are encountered, it is important to know how to approach making adjustments. In this case, it is necessary for you to exactly understand the physics of your contact problem and then apply what you are learning in this class to your model.
Geometric Simplification

Nonlinear contact can greatly increase the memory allocation, the number of iterative processes, and the computational time for a given model. Since the contact is the main focus of the analysis, it is usually pretty difficult to reduce the amount of contact in a given model without changing the physical behavior. One of the easiest ways to reduce the size of any simulation model is by simplifying the geometry. Any unnecessary complexities, i.e. holes, fillets, chamfers, should always be removed when possible. While these small features will certainly drive up the size of the model, the biggest reward can come from exposing symmetry and removing large chunks of geometry. Not all models will be symmetric, so it is important to be aware of when symmetry can and cannot be used appropriately.

For a symmetric model, symmetric boundary conditions can help:
(1) Reduce model size
(2) Decrease the model’s complexity
(3) Add stability

Another major area of simplification comes from further stabilizing unnecessary motion in the model. Adding contact to a model often adds additional freedom that is commonly overlooked as it may not be a major concern in the actual manufactured part, however, for the simulation it can be quite detrimental.

In the case of the “Static Stress with Nonlinear Material Models” analysis type, the model must be statically stable. If contact is added in such a way that the model is not statically stable, difficulty will be encountered when trying to reach a solution.

In the case of the “MES with Nonlinear Material Models” analysis type, models CAN have rigid body motion and are NOT required to be statically stable. However, it is important to be aware of the instabilities that exist in your model and if they are important to the physics of your simulation. For example, if you are analyzing the contact between a load carrying pin and a hole with frictionless contact, it is important to consider the rotation and translation of the pin that could occur along the axial direction. If this behavior is not important to your simulation, then constraining this negligible motion can improve the convergence of your solution. If these motions are not constrained, the simulation may spend more time trying to solve the spinning of the pin, rather than the load carrying interaction of the pin and the hole.
Choosing the Right Contact Definition Method – S2S vs. P2P

In MES nonlinear contact, contact pairs are commonly defined using two different approaches, surface to surface, or part to part. In either scenario, the determination of contact types and surface order (1st or 2nd) are also related to on the mesh size, surface shape of the contact surfaces. Since the contact interactions can sometimes be unknown, it is often more appealing to use the part to part contact method, as it provides the user with a much easier setup, not requiring the user to predict where contact will occur. It is important to realize that while the setup of this method is much easier from a user perspective, there are some disadvantages to be aware of, and that the recommended method for defining contact is surface to surface.

Disadvantages of part to part contact:

1. Increases model size
2. Increases run time and computing requirements
3. May lead to convergence problems due to inappropriate contact settings

Together, these disadvantages will almost always outweigh the ease of setup. We recommend using surface to surface contact for all contact pairs.

Choosing the Right Contact Updating Method – Automatic vs. Never

Before we discuss the contact updating scheme, we first need to understand what the contact updating scheme is actually doing, and that is a contact search. A contact search is one of the most important processes in a nonlinear contact analysis. It consists of two procedures: global search and local search. The global search finds a group of potentially active contact elements, and then the local search checks the exact status of each contact element. By use of global search, the memory allocation and the computation time can be reduced.

Contact updating is used to provide information to the global search process. “Never” and “Automatic” options are the two most commonly used options, where “Automatic” is the default.

The “Never” option only does a global search at very beginning of the analysis for each contact pair. In this case, the global search uses a maximum initial distance to limit the search region. Since the maximum initial distance for the contact search region can be defined by the user, it is best suited for problems with limited relative motion and allows the user to greatly reduce the size of the contact problem. In the case of a problem with large relative motion, the user would have to define a very large maximum initial distance to avoid cases of missing contact. This is not recommended as it can be quite computationally expensive.

The “Automatic” option determines whether a new global search is needed based on mesh size and relative motion. As large motion is encountered, the “Automatic” method will update the contact elements participating in the analysis. This updating will help keep the number of contact elements reasonable, as certain elements will be removed, and new ones will be added to adapt to new positions.

The control for the Contact Updating Method can be found on the General Surface to Surface dialog screen as seen in the following image.
Contact Stiffness

To discuss contact stiffness, we must notice that the penalty method is used to implement nonlinear contact, and contact stiffness (penalty stiffness) is used to enforce the compatibility in contact area.

In MES contact, there are two options to define contact stiffness: “Automatic” and “User-defined”. The “Automatic” option provides a value based on mesh size, material properties of contact parts, element types of contact parts, etc. This “Automatic” value can provide good convergence for most cases, but we may need to adjust the contact stiffness via the “user-defined” option for more challenging nonlinear contact models. In this section, we will focus our discussion on the “User-defined” contact stiffness option.
Contact stiffness is the most significant contact parameter. It influences both convergence and penetration. For a value that is too small, large penetration will occur, resulting in poor accuracy, however, the rate of convergence will be good. By increasing the contact stiffness, the user can prevent large penetration and improve accuracy at the contact surface, however, if the value is too large it will result in difficult convergence and poor efficiency. Now we encounter a dilemma: what is an appropriate value.

Here is a typical procedure for optimizing contact stiffness:

1. Estimate a trial value and run analysis:
   a. The “Automatic” option can be used as a start or
   b. Young’s Modulus/(10 to 100) is a good start if using “User-defined”

2. Examine the convergence and penetration

3. Adjust the value based on performance:
   a. Increase the value if large penetration happens
   b. Reduce the value if bad convergence appears

4. Rerun the analysis

**Frictional contact**

Frictional contact has always been plagued with more difficult convergence than frictionless contact. It has also been seen to show sliding occurring before the friction force exceeds static friction force. In this section, we will focus our discussion on how to improve the accuracy and convergence of frictional contact.

The application of friction in nonlinear contact is also via penalty method, and the calculation of friction force is shown in the following figure.
For better handling of the convergence and accuracy of frictional contact, a tangential ratio \( \text{Ratio}_{TS} \) is introduced to calculate the tangential contact stiffness

\[
F_f = K_f \cdot \alpha d_s
\]

Here is a typical procedure for optimizing the behavior of a friction contact pair:

1. Start with the default tangential stiffness ratio
   a. The default value of 1.0 is the biggest value accepted.
   b. A value of 0.01 is a good start.
2. Examine the convergence and relative sliding motion
3. Adjust the value
   a. Increase the value if large relative motion happens when friction force is small.
   b. Reduce the value if bad convergence appears.
4. Rerun the analysis
Setting up a Contact and Drop Test Model

Now that you have learned some basic fundamentals and techniques for working with contact models, we will look at the process for setting up a contact/drop test model. During this example, we will look at a scenario where a large container might be dropped while being unloaded from a flatbed trailer.

As with any simulation, we will need to start with a CAD geometry that represents the system we are interested in simulation. Once this model has been imported to Autodesk Simulation, we can begin defining all of the necessary details for performing an MES analysis with nonlinear contact. Typically, we would first start by meshing the model, however, when performing a contact analysis in MES with parts that start in contact, it is found to be beneficial to define the contact pairs prior to meshing. In the case of this example, the bottom of the container will be in contact with the top of the truck bed to start. Defining the contact pairs first will prevent mesh matching that occurs between mated parts during the meshing process. The mismatching of mesh in the contact pairs will improve solution convergence and improve run time.

As discussed, we will define our contact pairs using on canvas selections, rather than using the General Surface to Surface contact screen. We will need to consider where the container is initially in contact with the truck bed, and all of the areas it might impact when dropped. The following images represent some of these areas.
Container to truck bed:

Container to ground:

Container to wheels/tires:
Once the contact pairs are defined, we will proceed with meshing the model. For this model, our major concern is the container and how it behaves when dropped. We will look at using various element types, along with mixed modeling techniques to give us the most efficient solution possible. When preparing the CAD geometry for this simulation, the walls of the container were modeled as surfaces, which will allow us to use the plate/shell mesh type. A shell mesh will generate shell elements for the container walls as they are much too thin to accurately model with solid elements without drastically increasing the model size. We will use solid elements for the remainder of the model. Specifically, we will use 3D Kinematic elements for the truck itself, as we are not concerned with the results in those components. The 3D Kinematic elements do not undergo a stress calculation, further increasing the efficiency in our solution.

With the meshing and part definition process complete, we will look at defining loads and boundary conditions to represent the behavior of lifting the container from the truck and having it drop before being lowered to the ground. The use of prescribed displacements will allow us to do just this. Prescribed displacements allow the user to define a direction of motion, along with a load curve that will define the rate of motion. Additionally, prescribed displacements have the ability to be instantly turned off which will allow us to represent the "dropping" of the container. This model requires two sets of prescribed displacements, one for the vertical lifting motion, and one set for the horizontal motion to move the container away from the truck and they will be applied to the top 4 corner nodes of the container.
Advanced Techniques for Nonlinear Contact and Drop Shock Analysis

With the motion and event duration defined, our last point of focus will be on the contact settings for this model. As we have learned in this class, the right contact settings can make or break an analysis. Since this is a drop test, we will define the contact problem type as High Speed. We will also define the contact method as Frictional to best represent how the container will interact with the ground. To prevent any issues with penetration due to the order of the contact surfaces, we will choose the contact type to be surface to surface. *Note: This will be more computationally expensive than using point to surface.* Our last step will be to define the friction information and the contact stiffness. As you can see in the figure below, static and sliding friction coefficients have been defined based on assumptions made for the friction between the parts. Based on our learning above, we have decided to define a tangential stiffness ratio of 0.01 to get the most efficient calculation.
With the friction parameters defined, our last order of business will be to define contact stiffness. In the case of this model, rather than using automatic, we chose to define a “user defined contact stiffness.” As we noted, a good rule of thumb for contact stiffness is Young’s Modulus/10 to 100. In the case of this model, we are looking at steel (E=29e6 psi) as the material for the bottom rails of the container and we chose to use a value of 29e4 as the contact stiffness. With the contact settings defined, the model is ready to run.

Closing Thoughts…

In summary here a few tips to keep handy when working with nonlinear contact models in Autodesk Simulation.

- Use MES for nonlinear contact analyses
- Use “Automatic” when defining the Contact Type
- Use “Large displacement” for MES analysis type
- Monitor convergence via level and residual feedback
- Simplify geometry where possible
- Stabilize models to prevent unnecessary motion
- Use “Never” updating option when small relative motion is present
- Use the trial-and-error procedure to adjust contact stiffness
- Reduce tangential stiffness ratio to 0.01

If you have any further questions regarding topics covered in the class, please contact Shoubing Zhuang or Mike Smell via the AU website, or at simssquad@autodesk.com.