Connections (Linear Contact)

A Seminar for FEMAP v11.2.2 Users

Adrian Jensen, PE | Senior Staff Mechanical Engineer
Kyle Hamilton | Staff Mechanical Engineer

FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA Sales, Support & Consulting

www.PredictiveEngineering.com

AppliedCAx

Siemens PLM Software Sales & Support
CAD | CAM | CAE | Teamcenter

www.AppliedCAx.com

Table of Contents

1. INTRODUCTION

Fatal errors and/or crazy stresses!

One of the most common topics for our technical support team is Connections (also known as linear contact). You've seen some great demos, you've got the perfect pin/clevis model and you found the "Automatic" button under the Connect menu. However, once your computer finally stops spinning, the analysis crashes. Sound familiar? The goal of this seminar is break Connections down to the simple components and provide a basic understanding of how they work. That, combined with some suggested default setting should have you up-and-running for your next contact analysis.

But isn't contact nonlinear by nature?

Linear contact is called such because there is no geometric or material non-linearity in the analysis. That is, the stiffness matrix is not updated and the material behaves in a linear elastic fashion. Take for example a small steel punch pressing against a rigid surface. Let's assume it's not going to buckle and it has nominal material properties for steel. When the steel reaches the yield stress, it has a strain of 0.2% (60,000 psi yield / 29,000,000 psi modulus). If that punch was originally 1" long, it has compressed 0.002". One could safely assume that the behavior of the system has changed very little (it's not buckling, remember?) What does this all mean? For problems with stiff engineering materials (e.g. steel, aluminum, laminates) where the material is not stressed beyond the yield strength, linear contact is a reasonable analysis mechanism.

**Just tell me what buttons to push...**

This white paper will cover a wide range of linear contact problems, from the simple task of setting up contact regions to the more daunting charge of manually adjusting the penalty factors. The key to any linear contact problem is to start simple. Before you try to run that 500,000 node model with 20 contact regions, build small, manageable models of the different contact scenarios for easy debugging.

2. CONNECTION REGIONS

When you break a linear contact problem down into its most simple component, you have a Connection Region. It might seem like an elementary step in the analysis but if it is not done correctly, there will be problems that can't be fixed by toying with the more advanced components of linear contact.

2.1 CONNECTION REGIONS: SOLIDS, BY SURFACES

The easiest linear contact analysis to set up is for a simple solid model using surfaces. When creating your Connection Region by surfaces you'll notice an option to select the positive or negative. For solid models, this option is not necessary. If you check the normals of the surfaces of the solid body, they're all facing outward. The option to modify surface normals only works for sheet solids.
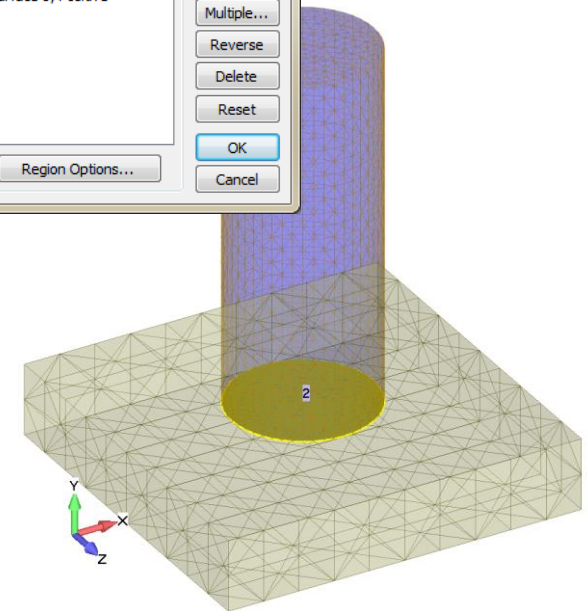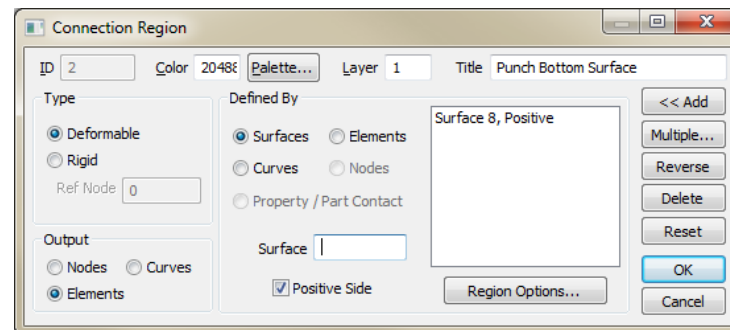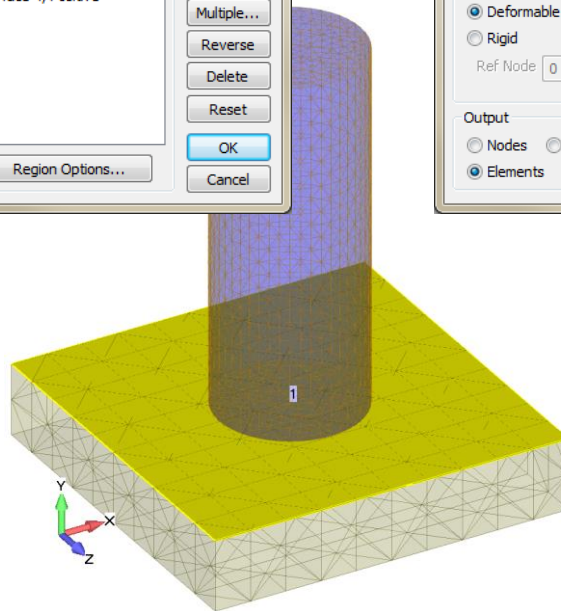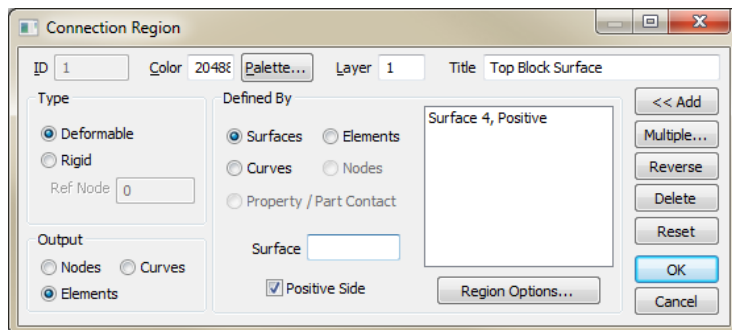
2.2   CONNECTION REGIONS: SOLIDS, BY ELEMENTS

In some cases you be unable to select your Connection Regions by surfaces. The other option is to select your regions by elements. This technique gives you more control over selecting your Connection Region, but also opens up the possibility for error. Start to create you Connection Region by selecting the "Elements" radio button under the "Defined By" section of the Connection Region dialogue box.

Assuming you don't want to select elements one-by-one, click the "Multiple" button. There are two stages to the region selection by elements: the element selection and the element face selection. The element selection is simple; grab all elements that have faces you want to include in the Connection Region.
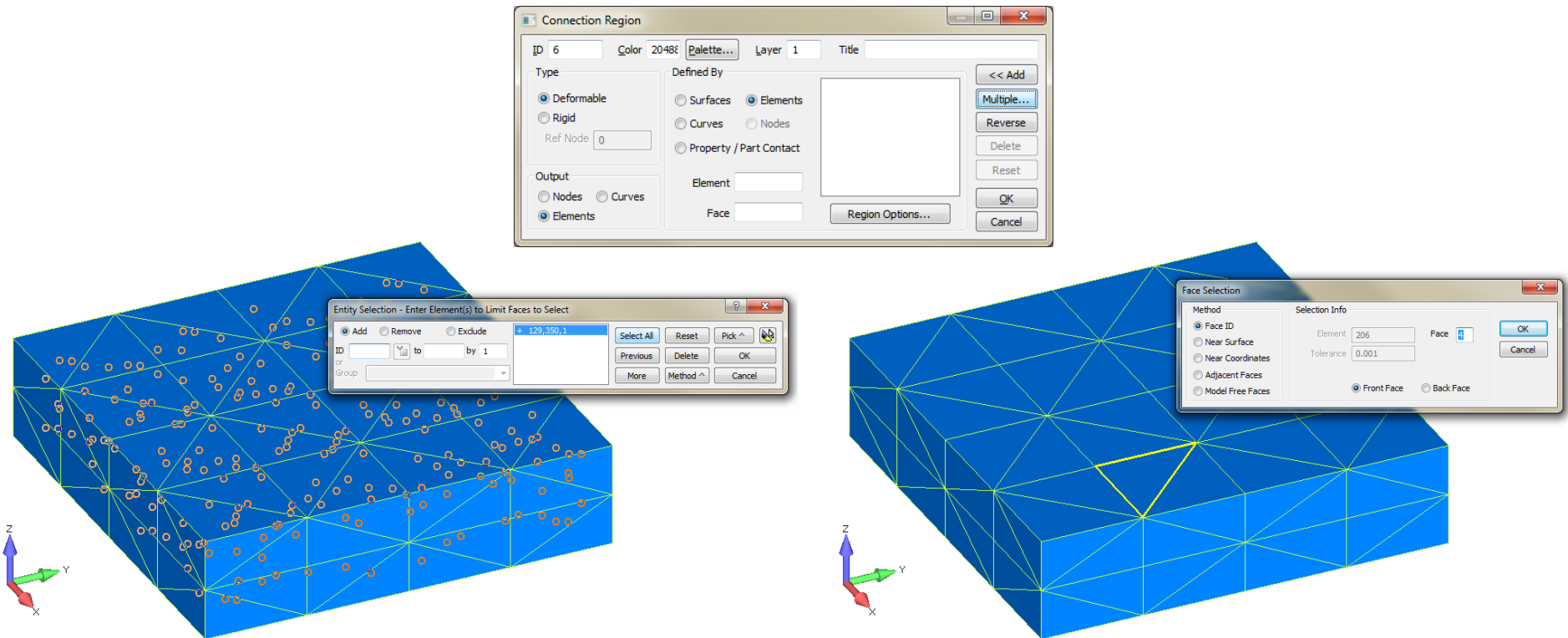
The element face selection is a bit trickier. Once you have selected the elements you want, you are presented five options for face selection. The following pages provide examples of each.

Face ID

Each face of an element has a face id. Plate elements use faces 1 & 2, tet elements use faces 1-4 and hex elements use faces 1-6. This option is generally the least useful for a solid model because the element faces on the surface of the solid do not necessarily share a common face ID. The graphics below show the Connection Regions created by selecting all of the elements and using the "Face ID" face selection method. The first graphic shows face 1, the second shows face 2 and so forth.
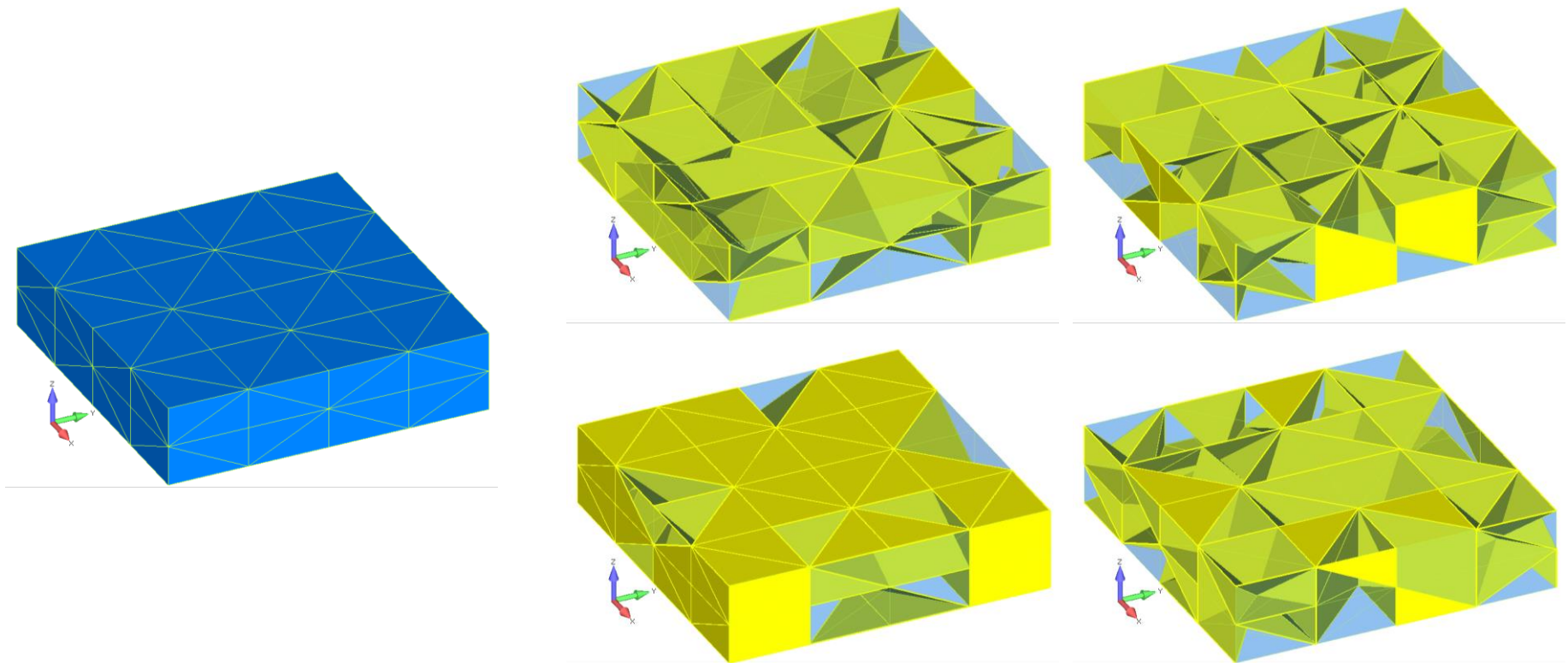
Near Surface

This option is useful if you have geometry to work with but you don't want to limit the elements in your Connection Region. The graphics below show the Connection Regions created by selecting a cluster of elements in the corner of the block and using the "Near Surface" face selection method.
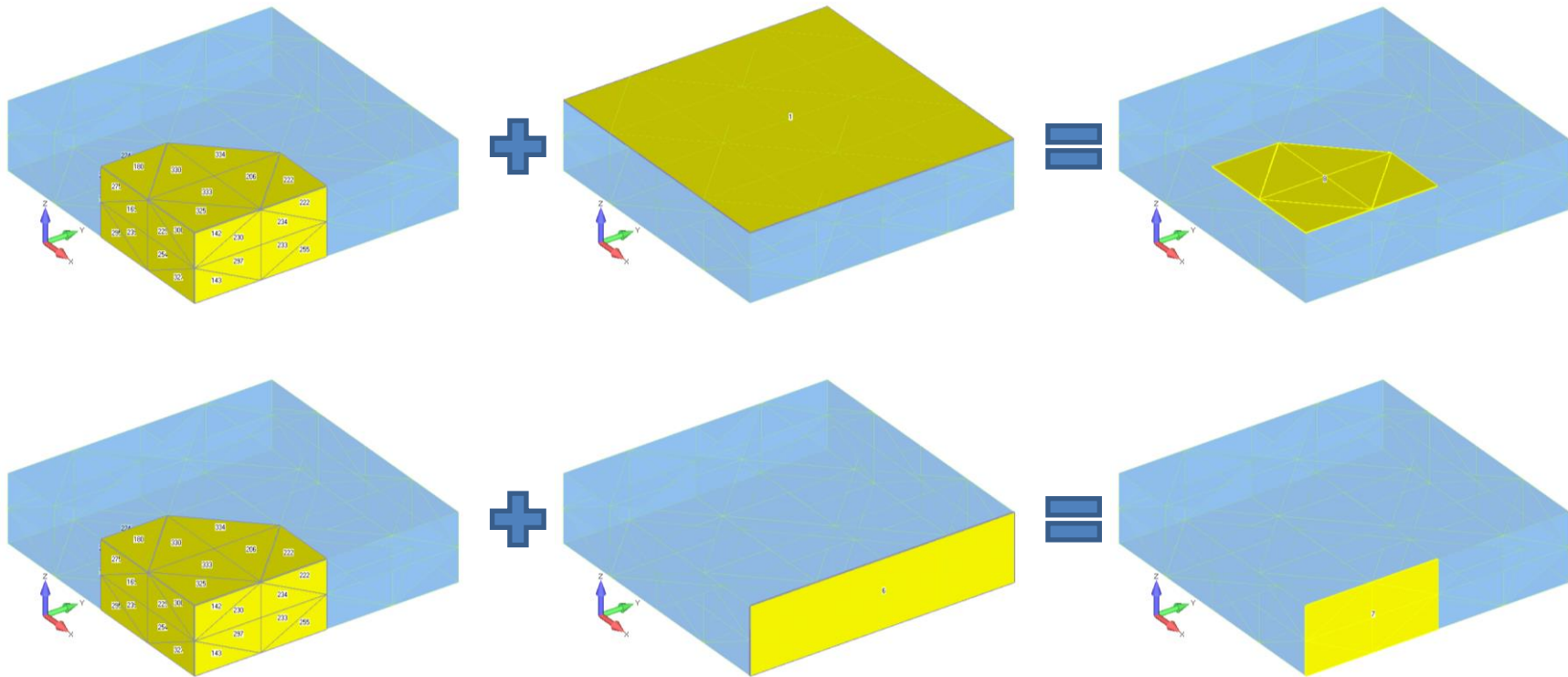

Near Coordinates

The "Near Coordinates" method is very similar to "Near Surface". Instead of specifying a surface, however, you choose a coordinate system, direction and position. This defines a planar surface, which is used along with the tolerance to find the closest faces.

Adjacent Faces

A powerful method for choosing faces, especially for complex solid and planar element models. You choose just one initial face (and the associated element ID). This can be done very easily by graphically selecting the face. You then specify a tolerance angle. FEMAP will search all selected elements for faces that are connected to the face that you chose and that are within the specified tolerance from being coplanar with an already selected face.
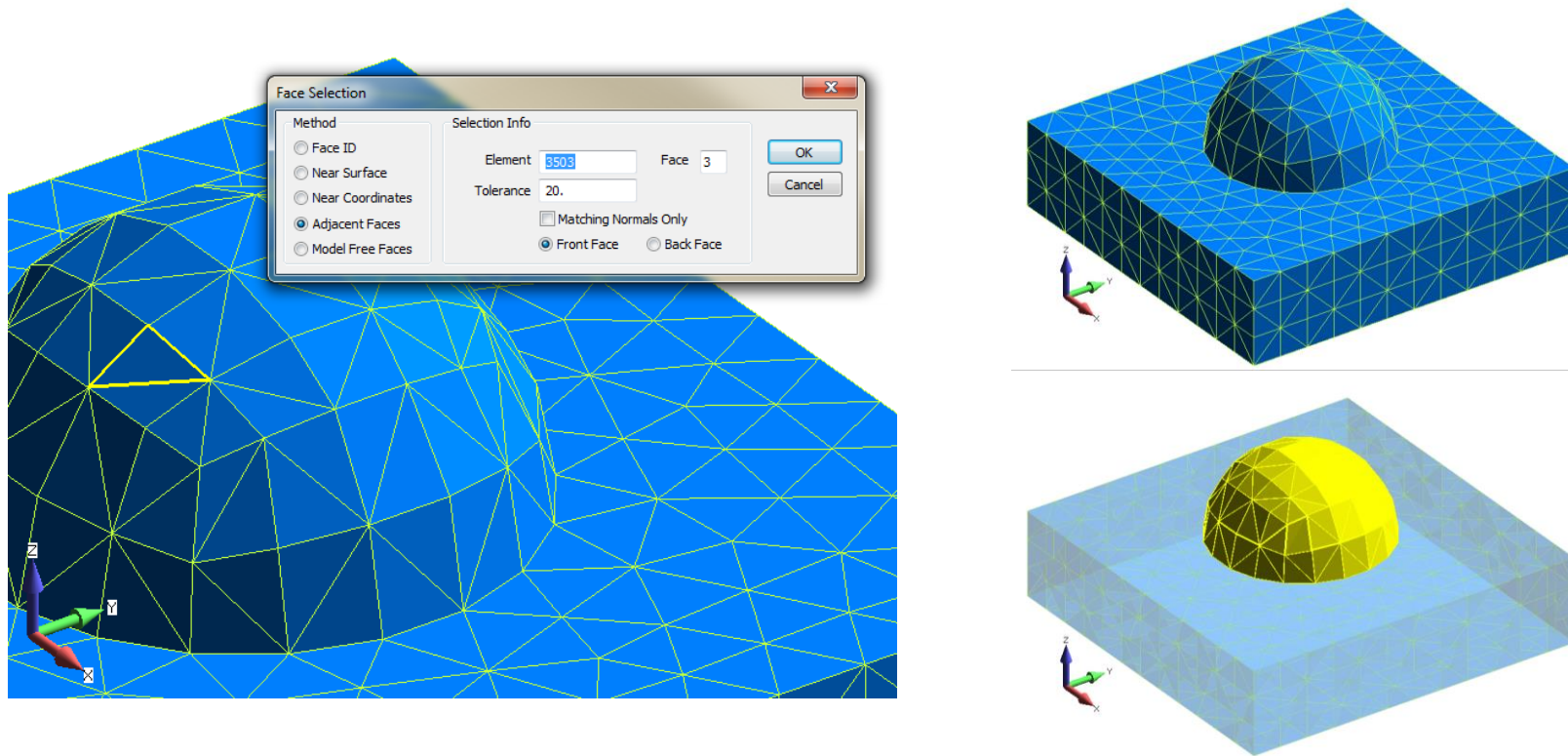
Model Free Faces

The "Model Free Faces" method simply includes every "free element face" in your element selection. A good technique if you can grab exactly the elements you want within the element selection dialogue box.
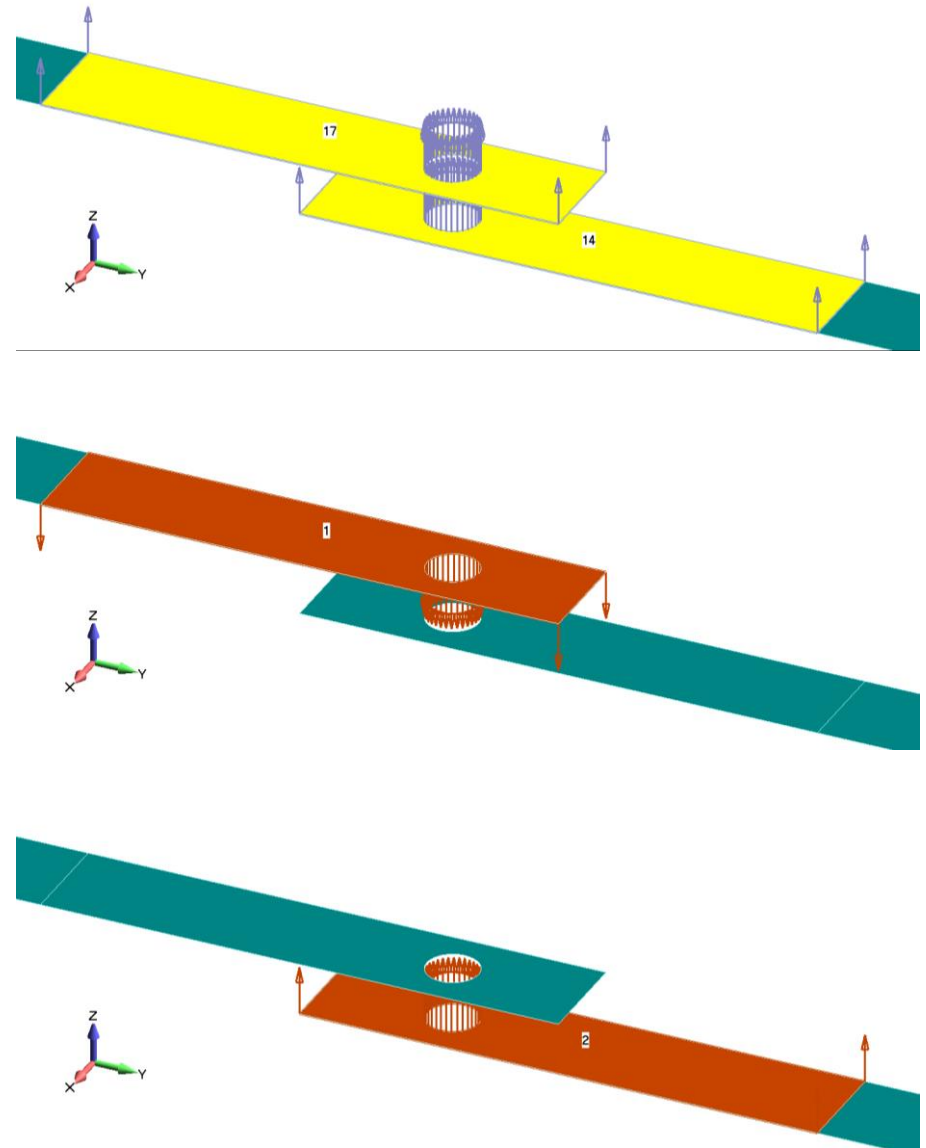
2.3 CONNECTION REGIONS: PLATES, BY SURFACES

Plate models generally require more care when selecting your regions. When selecting you plate model Connection Regions by surfaces, you need to pay special attention to the surface normals and element normals. Since both faces of a plate element are linked to the same surface, you will need to turn the "Positive Side" option on and off where appropriate.

The surface normals are shown in the upper-most graphic on the right. The middle image shows the first Connection Region. The surface was selected with the "Positive Side" option unchecked. The bottom image shows the second Connection Region. The surface was selected with the "Positive Side" option checked.

Note that the Connection Region directions face each other in a properly configured contact analysis. You can check this by using Window > Show Entities. Select "Region" for your entity type and check the "Show Normals" style option.

2.4 CONNECTION REGIONS: PLATES, BY ELEMENTS

In some cases you may be unable to select your Connection Regions by surfaces. The other option is to select your regions by elements. This technique gives you more control over selecting your Connection Region, but also opens up the possibility for error. Start to create you Connection Region by selecting the "Elements" radio button under the "Defined By" section of the Connection Region dialogue box. You probably don't want to select elements one-by-one. Use the "Multiple" button to speed things up.
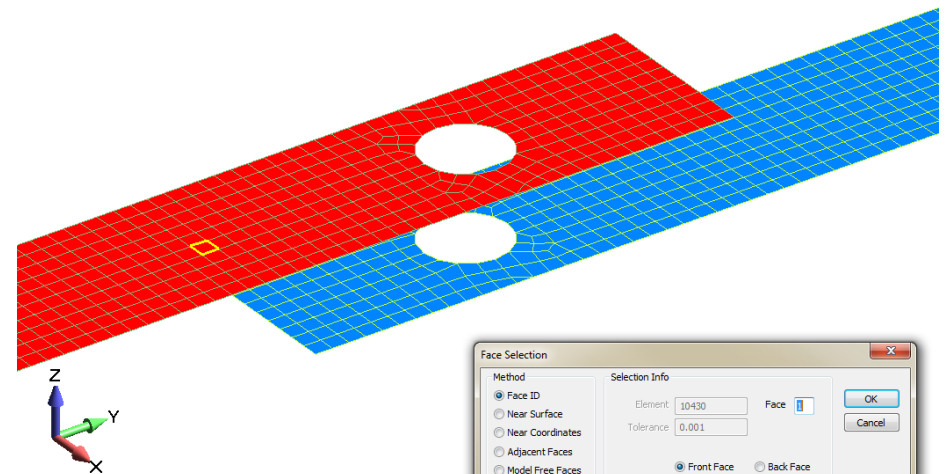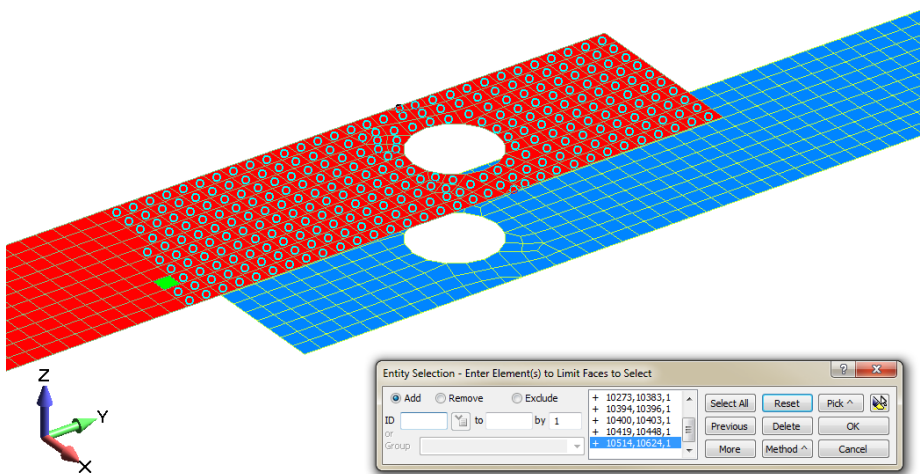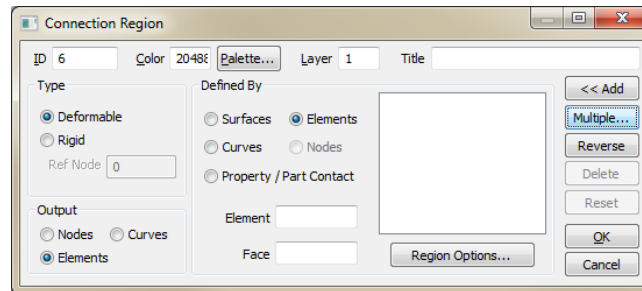
Once you have selected the elements you want, you are presented five options for face selection. When working with plate elements, only the "Face ID" and "Adjacent Faces" options should be used.

Face ID

As mentioned earlier, plate elements use faces 1 & 2 for Connection Regions. Be aware that a four sided plate element has six faces and Femap will allow you to select any of these faces but only faces 1 & 2 are supported for contact. The "Face ID" method allows you to enter the face ID numerically or select a face from the model. This model is more effective with plate models because the faces of the elements are more commonly aligned.

Adjacent Faces

This method is useful for creating Connection Regions when elements do not have consistent orientation or when the model has complex geometry (see "Adjacent Faces" in the solid element section).
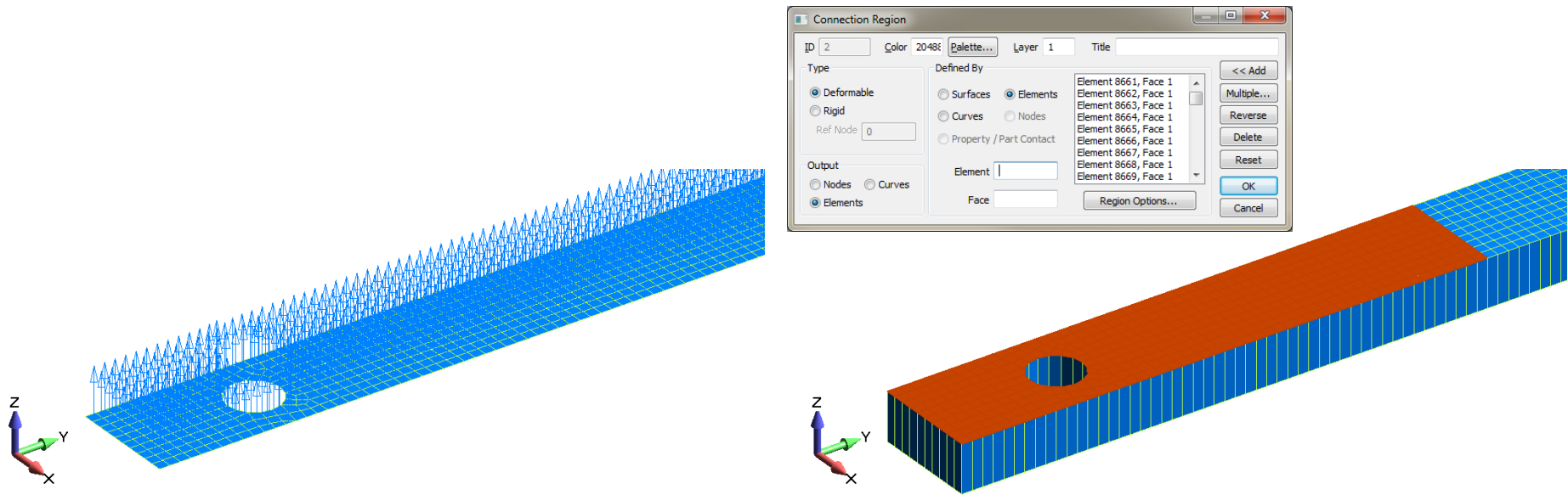
3. CONNECTORS

Once you have properly set up your Connection Regions, you must establish contact pairs, called "Connectors" in the Femap interface. A Connector specifies a contact mechanism in the model with a "Source" Connection Region, a "Target" Connection Region and a Connection Property.
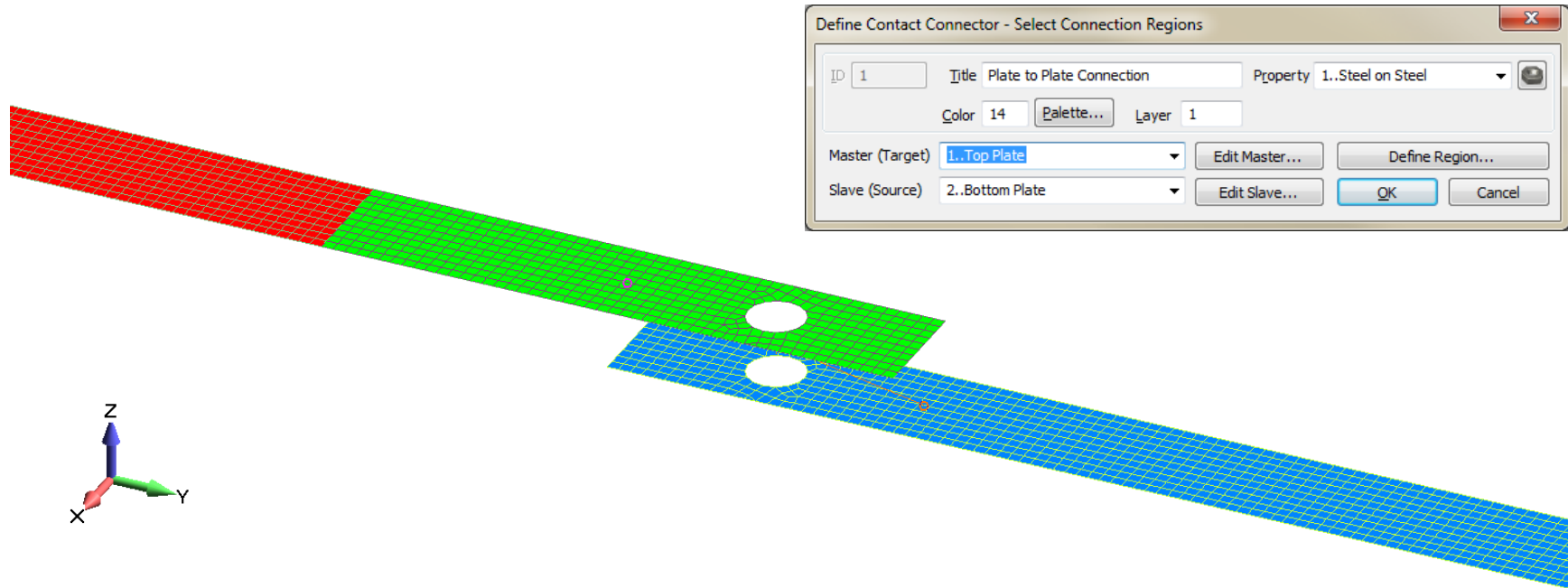
Setting up a Connector is quite simple, select a Connection Property from the drop-down list or use the quick button to the right to create a new one. Leave the fields of the Connection Property blank for now. Connection Properties will be discussed in greater detail later in this white paper. Next, select Master (Target) and Slave (Source) Connection Regions from the drop-down lists or select by clicking on the desired Connection Regions in the model. The Connection Regions will highlight as you place the cursor over them to make the selection process easier (see the graphic above).

3.1   CONNECTORS: SOURCE REGIONS AND TARGET REGIONS, FROM THE NX NASTRAN USER'S GUIDE

It's important to understand how contact element are created when selecting which region will be the source and which the target, since the two can be interchangeable. The solver projects vector normals from the source region to the target region. It then creates contact elements when these normals intersect elements in the target region and are within the search distance criteria for the contact pair. This means that when the two regions of a pair do not have corresponding one-to-one elements, the number of contact elements that the solver creates can change depending on which region it projects the elements from and which region it projects them to.

In general, of the two contact regions you use for the pair, choose the one with the finer mesh for the source region. When the source and target regions have different mesh densities, more elements on the source region will mean that more contact elements are created, which will produce a more accurate solution.

For example, the two regions below are both composed of linear shells. The source region (A) has one element and the target region (B) has four elements.

When creating the contact elements between these regions, the software projects contact elements from the single element on the source region to the four elements in the target region. This results in the creation of a single contact element (see the graphic on the left).

However, if you were to use the region with four elements as the source (C) and the region with one element as the target (D), the solution will create 4 contact elements (see the graphic on the right).

In the example below, both the sphere and the rectangular plate were meshed with plate elements. The sphere has a higher mesh density. The edges of the rectangle are fixed and the sphere is pushed downward into the plate.
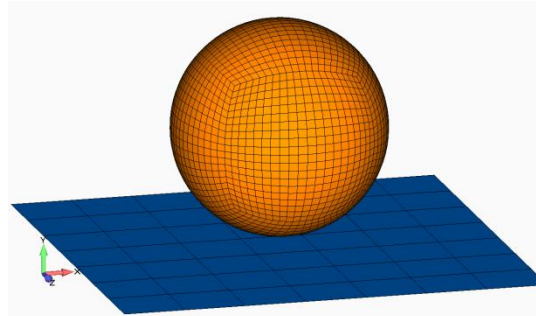
In the graphics below, the sphere is viewed from the bottom with contact pressure contoured over the model. In the graphic on the left, the Connector was setup with the sphere as the Source Region and the r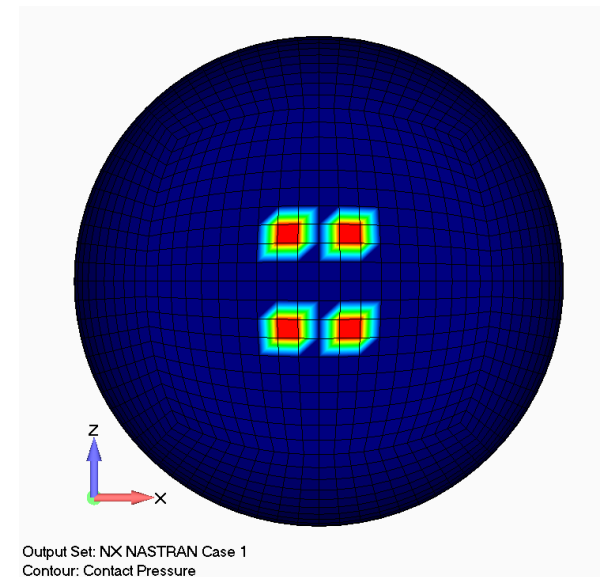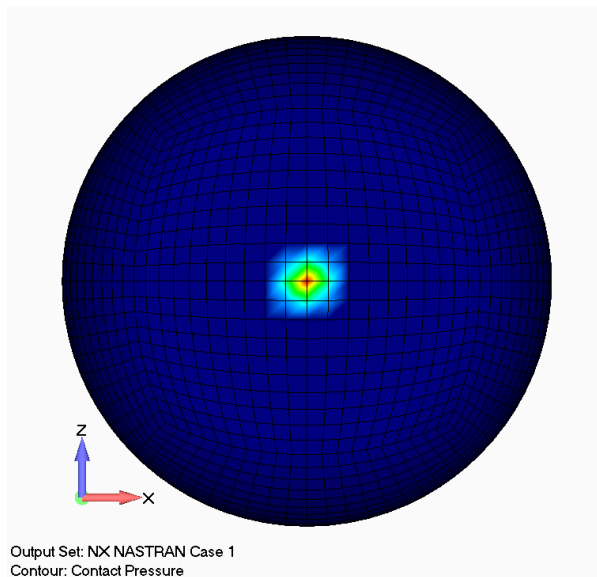ectangular plate as the Target Region. As predicted, a single hot spot is present at the bottom-center of the sphere. In the graphic on the right, the Connector was setup with the rectangular plate as the Source Region and the sphere as the Target Region. Four separate hot spots are present, each covering four corners of an element. This result set is clearly incorrect and is caused by a lack of contact elements created for the analysis.

Output Set: NX NASTRAN Case 1
Contour: Contact Pressure

Output Set: NX NASTRAN Case 1
Contour: Contact Pressure

FINITE ELEMENT ANALYSIS
Predictive Engineering

Connection Properties

After you have created and checked your Connection Regions and Connectors you can set up the Connection Property.

Although a Connection Property is required to create a Connector, it is recommended that fully configuring the Connection Property should be saved for last. This is because it is easy to get overwhelmed with the options provided on a Connection Property card. Additionally, unless the Connection Regions and Connectors are configured correctly, none of the options of the Connection Property will allow for a accurate contact analysis.

The Connection Property card is divided into three sections: Contact Pair (BCTSET), Contact Property (BCTPARAM) and Common Contact Parameters (BCTPARAM). Common Glue Parameters (BGPARAM) and Glued Contact Property (BGSET) will not be covered in this white paper.
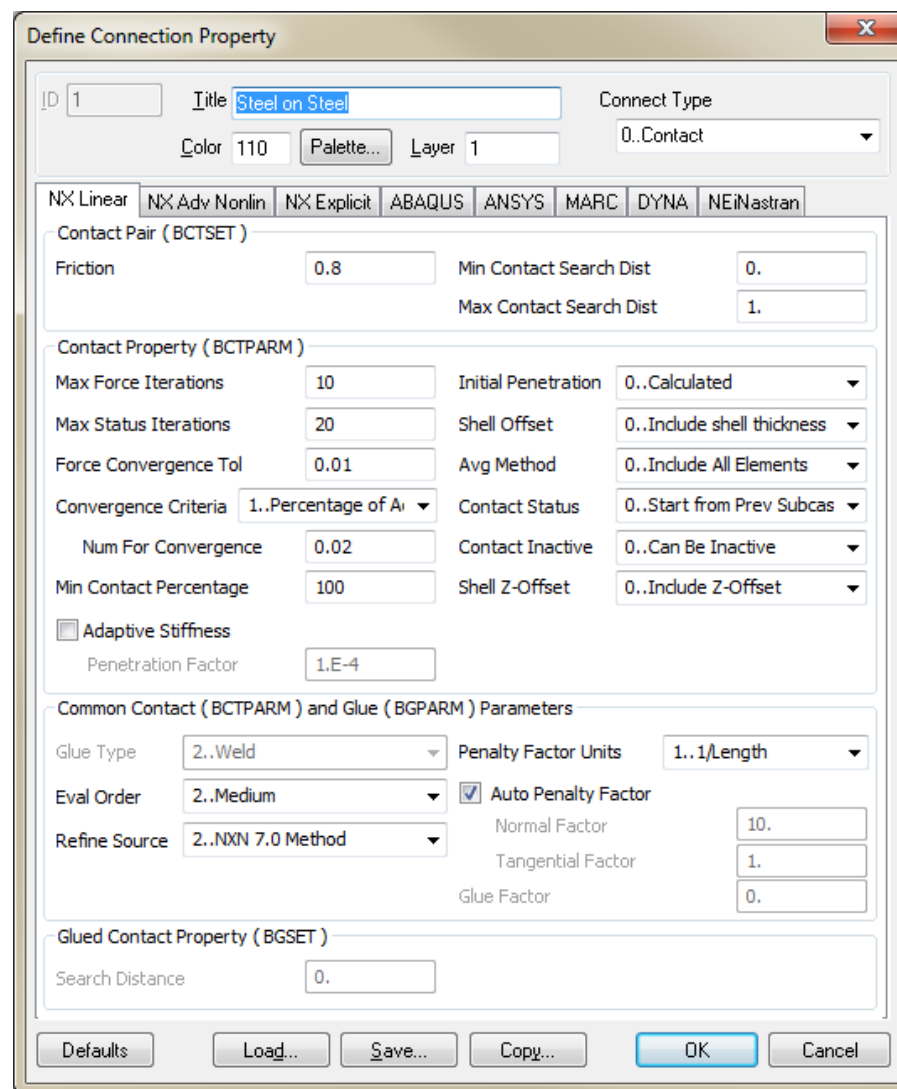
3.2   CONNECTION PROPERTIES: CONTACT PAIR (BCTSET)

The options in this portion of the dialog box can be set individually for each Connector (contact pair) that is created in the model. These options will be written out to the BCTSET entry for each individual contact pair. Each contact pair will be designated in the graphics window with a single line going from one Connection Region to another and this line is a contact element.

Friction

The static coefficient of friction for the contact pair. For many contact analyses it is important to include a Friction value to provide stability to the model. For example, in the image to the right, a small, arbitrary Friction value was used to constrain the inner tube from translating in the Z direction.

Note: In general, if different friction values are NOT needed then the contact pairs should all reference the same contact property.
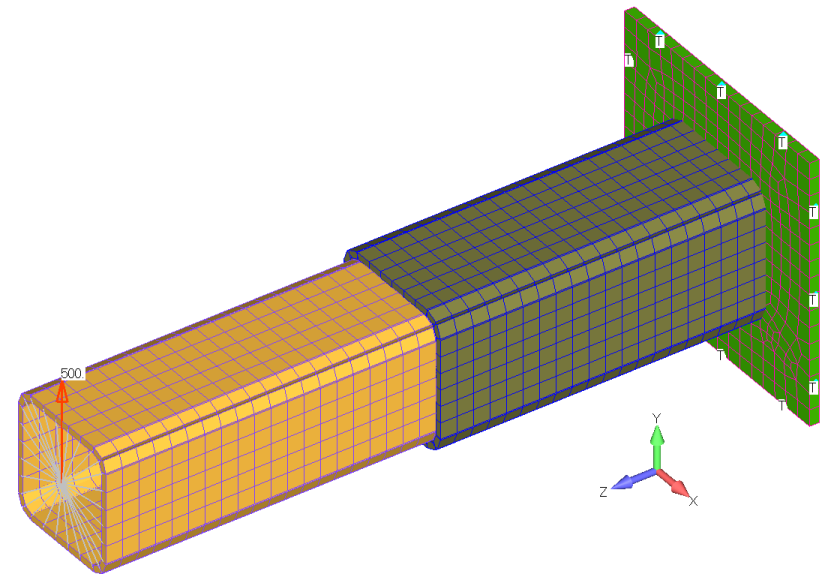
Min Contact Search Dist/Max Contact Search Distance

As discussed in the Connectors section, contact elements are created when these normals projected from the source region intersect elements in the target region within the search distance. The search distance is specified with Min Contact Search Dist and Max Contact Search Dist.

Note: The minimum distance can be negative and used for an interference fit condition modeled as overlapping surfaces.

3.3 CONNECTION PROPERTIES: CONTACT PROPERTY (BCTPARM)

Initial Penetration

Controls how Nastran handles initial gap or penetration of the generated contact elements. This setting is particularly useful if some of your elements unintentionally penetrate each other and you do not wish to modify or rebuild your model.

> 0..Calculated - Use the value calculated from the grid coordinates. In the case of penetrations, a model may experience "press fit" behavior when using this option.
>
> 2..Calculated/Zero Penetrations - Same as 0..Calculated, but if penetration is detected, set the value to zero.
>
> 3..Zero Gap/Penetration - Sets the penetration/gap to zero for all contact elements.
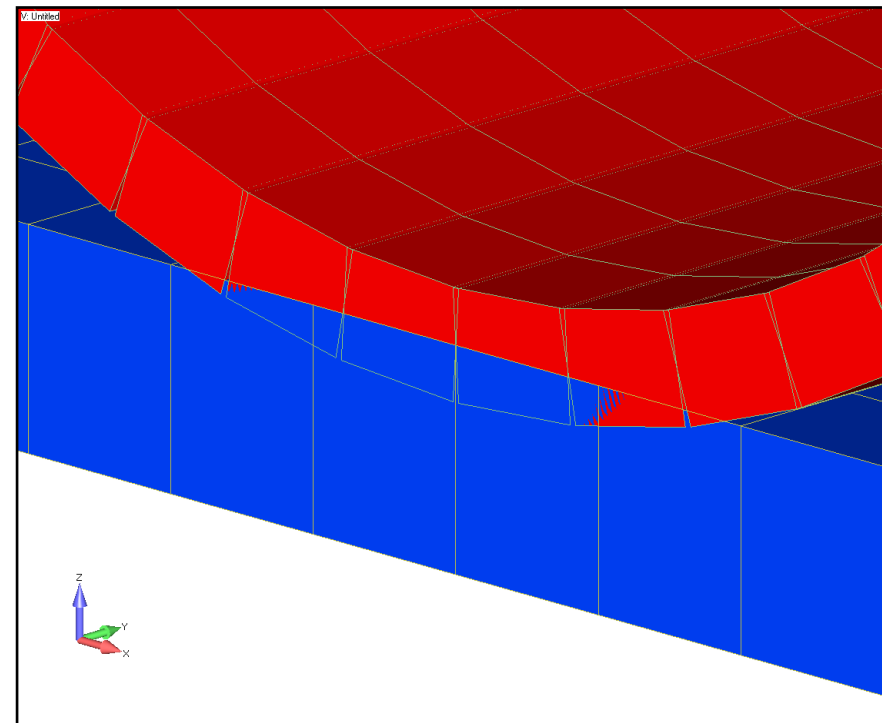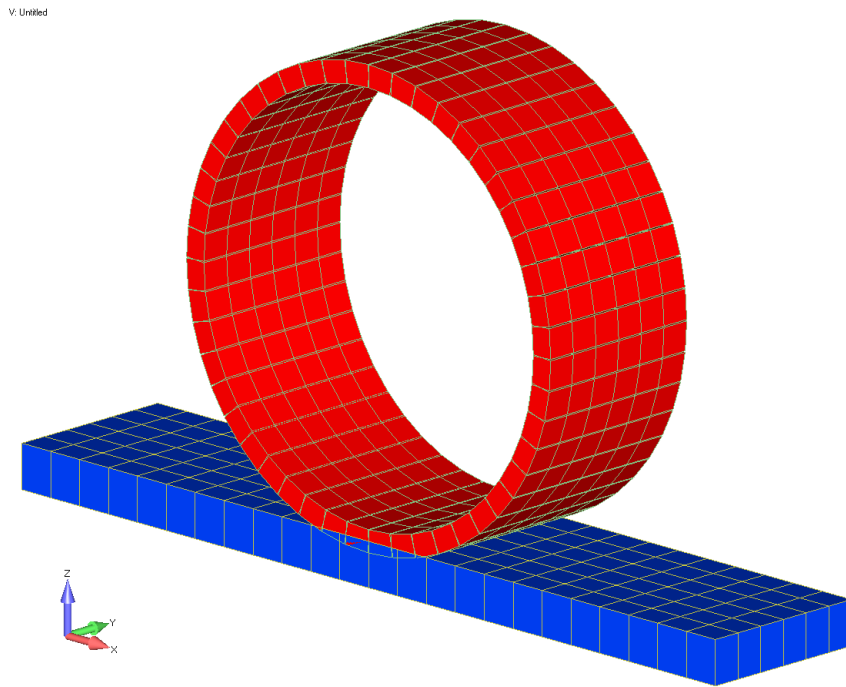
Shell Offset

For plate elements, you can enforce contact at either the midplane or the surface. This option, as well as the Initial Penetration option, can save modeling time when optimizing a plate thickness of a component in a contact region. The gap between connection regions will be independent of plate element thickness if this option is turned off.

0..Include shell thickness - Include half shell thickness as surface offset.

1..Do not include thickness - Does not include thickness offset.

The images on the left show results with Shell Offset turned off and the images to the right show results with the Shell Offset option turned on. The models are displayed with and without thickness on top and bottom respectively.

Output Set: Shell Offset - OFF
Deformed(1.728): Total Translation

Output Set: Shell Offset - ON
Deformed(1.478): Total Translation

Output Set: Shell Offset - OFF
Deformed(1.728): Total Translation

Output Set: Shell Offset - ON
Deformed(1.478): Total Translation
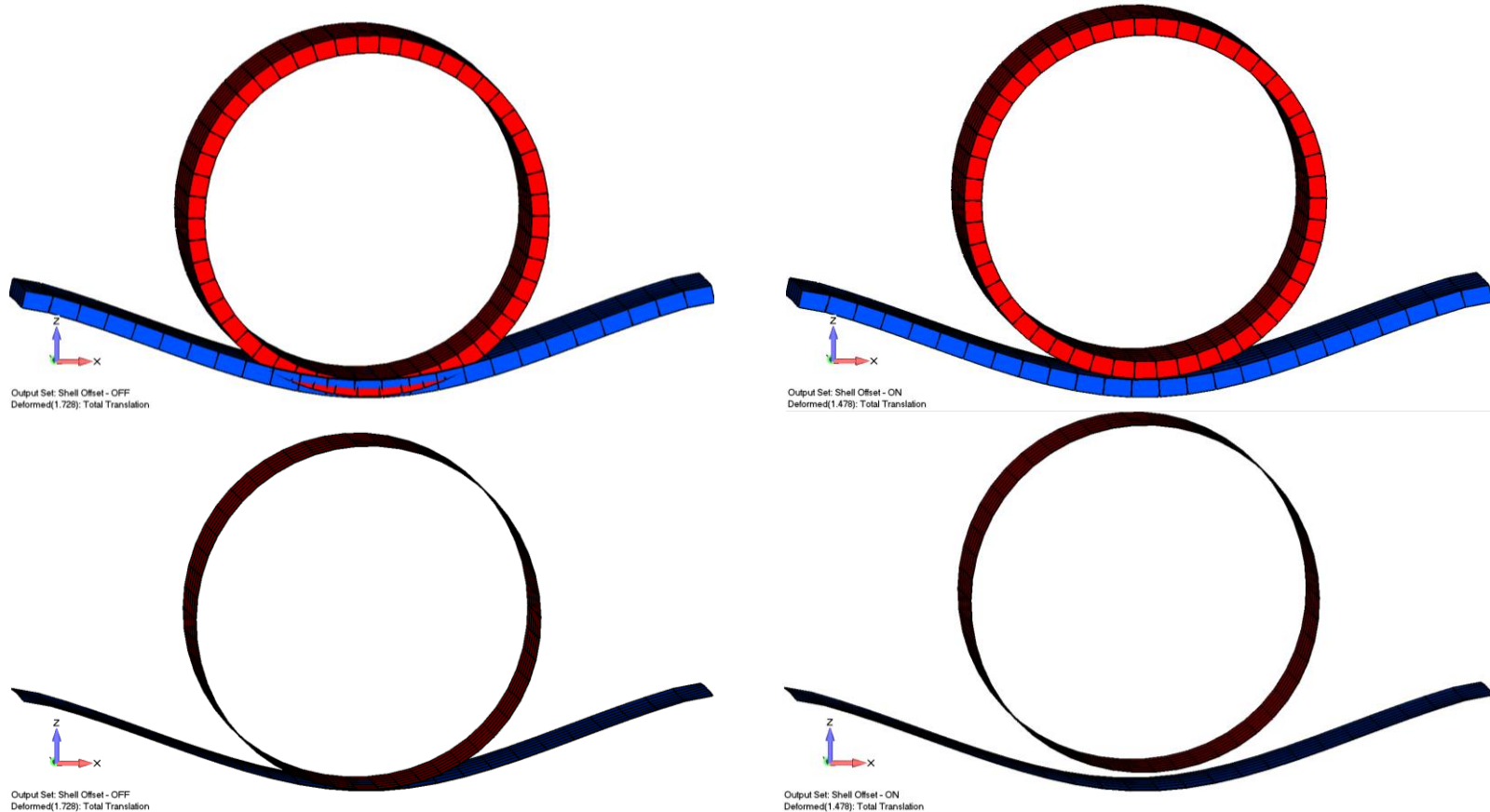
Shell Z-Offset

Allows you to choose if the Z-Offset on shell elements should be included in the contact analysis.

0..Include Z-Offset (Default) - Z offset of shells is included as surface offset.

1..No not Include Z-Offset - Z offset of shells is NOT included as surface offset.

The images on the left show results with Shell Z Offset turned off and the images to the right show results with the Shell Z Offset option turned on. The models are displayed with and without element offsets on top and bottom respectively.
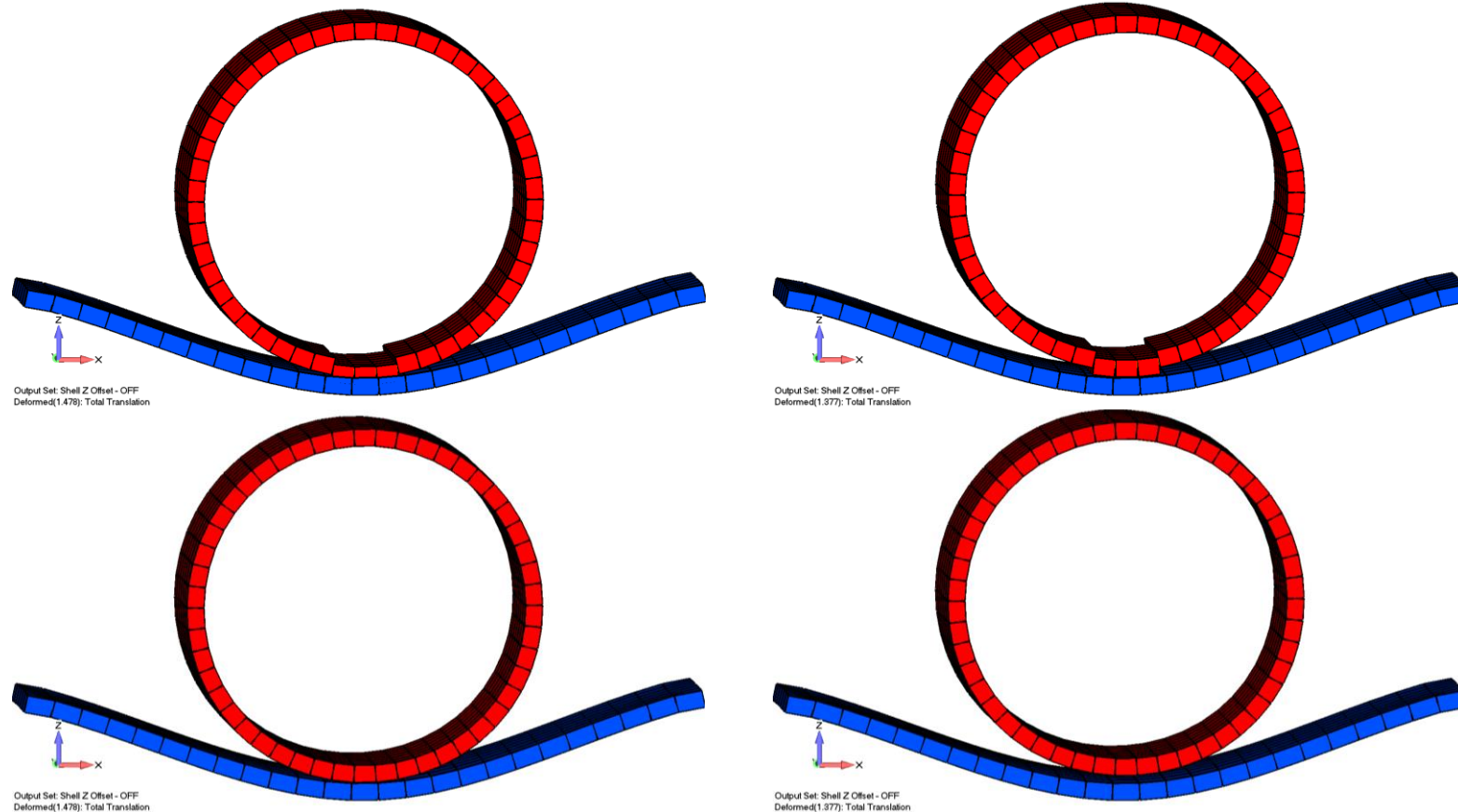
These options need to only be defined once for a contact analysis, regardless of how many *Connectors* (contact pairs) are defined in the model. Each *Connector* has an ID assigned to it and can reference a different *Connection Property*. FEMAP will use the *Connection Property* referenced by the *Connector* with the lowest ID to define the BCTPARM entry for the entire model. For example, if a model has 2 *Connectors* (contact pairs) with ID numbers 101 and 102, the *Connection Property* values defined in the property associated with *Connector* ID 101 would be used for the analysis.

Max Force Iterations - Creates the MAXF field on the BCTPARM entry. Designates the maximum number of iterations for a force (inner) loop (Default = 10).

Max Status Iterations - Creates the MAXS field on the BCTPARM entry. Designates the maximum number of iterations for a status (outer) loop (Default = 20).

Force Convergence Tol - Creates the CTOL field on the BCTPARM entry. Designates the Contact Force convergence tolerance (Default = 0.01).

Convergence Criteria and Num For Convergence - Together, these two values create the NCHG field on the BCTPARM entry. The value and type of number (real or integer) entered for *Num For Convergence* depends on the option set for *Convergence Criteria*:

> 0..Number of Changes - When this option is set, *Num for Convergence* must be an integer > 1. This value defines the allowable number of contact changes.

> 1..Percentage of Active - When this option is set, *Num for Convergence* must be entered as a percentage (between 1 and 99, which will appear as 0.01 to 0.99 in the NX Nastran input file). The solver treats this value as a percentage of the number of active contact elements in each outer loop of the contact algorithm. The number of active contact elements is evaluated at each outer loop iteration.

Min Contact Percentage - Creates the MPER field on the BCTPARM entry. Designates the Minimum Contact Set Percentage (Default = 100).

Adaptive Stiffness - This is a flag to indicate whether adaptive stiffness is activated. Creates PENADAPT field on BCTPARM entry (Default=0). When not checked, it places a "0" (No Adaptive adjustment) into the PENADAPT field, when checked places a "1" (Adaptively adjusts contact stiffness) into the PENADAPT field.

Penetration Factor - Creates the PENETFAC field on the BCTPARM entry (Default = 1.0E-4). Designates the penetration factor for adaptive penalty stiffness adjustment. Only used when Adaptive Stiffness is "on" and should usually only be set to a lower value to reduce the amount of penetration allowed to occur in an analysis.

Avg Methods - Creates the AVGSTS field on the BCTPARM entry. Determines the averaging method for contact pressure/traction results (Default = 0).

> 0..Include All Elements - The averaging of Pressure/Traction values for a contact grid will include the results from ALL contact elements attached to the grid regardless of whether they are active or inactive in the contact problem

> 1..Include Active Elements - The averaging of the Pressure/Traction values for a contact grid will exclude those contact elements which are not active in the contact solution and thus have a zero Pressure/Traction value.

Contact Status - Creates the RESET field on the BCTPARM entry. Flag to indicate if the contact status for a specific subcase is to start from the final status of the previous subcase. (Default =0)

> 0..Start from Prev Subcase - Starts from previous subcase.

> 1..Start from Init State- Starts from initial state.

Contact Inactive - Creates the CSTRAT field on the BCTPARM entry. When set to "1..Restrict From Inactive", prevents all of the contact elements from becoming inactive.

    0..Can Be Inactive (Default) - All contact elements can become inactive

    1..Restrict From Inactive - Solver will reduce the likelihood of all of the contact elements becoming inactive.

Note: Under certain conditions, all of the contact elements could become inactive which may lead to singularities. Setting the parameter to "1..Restrict From Inactive" will reduce the possibility of this happening.

Eval Order - Determines the number of "Linear Contact Points" for a single element on the source region. As discussed in the Connectors section, the solver projects vector normals from the source region to the target region. You can increase the density of these vectors without re-meshing by increasing your evaluation order. A guide to the "Linear Contact Points" per element is provided below.

    0..Default - Simply uses the default value for "Linear" or "Glued" contact built into the NX Nastran solver.

    1..Low - Lowest order of points on source region.

    2..Medium - Medium order of points on source region. This is the default.

    3..High - Highest order of points on source region.

The higher the integration order, the longer the solve will take.

Refine Source - Determines if the source region is refined and what refinement method will be used for the Contact solution.

>   0..Do Not Refine - Does not refine the "Linear Contact" source region based on target surface definition.

>   1..Refine Source to Target - Refines the "Linear Contact/Glue" source region based on target surface definition.

>   2..NXN 7.0 Method - Refines the "Linear Contact/Glue" source region using the NX Nastran 7.0 method.

| Face Type	Contact Evaluation Order		
	1..Low	2..Medium	3..High
Linear Triangle	1	3	7
Parabolic Triangle	3	7	12
Linear Quad	1	4	9
Parabolic Quad	4	9	16

In the Connector section of this White Paper, an example was presented showing the effects of choosing a Source/Target with fine/course mesh density. For that example, Refine Source option 0..Do Not Refine, was used. A graphic from this analysis is shown on the left.

If we analyze the same model (with the coarsely meshed rectangular plate Connector Source) but change the Refine Source option to 2..NXN 7.0 Method, we see much better results. A graphic from this analysis is shown on the right.
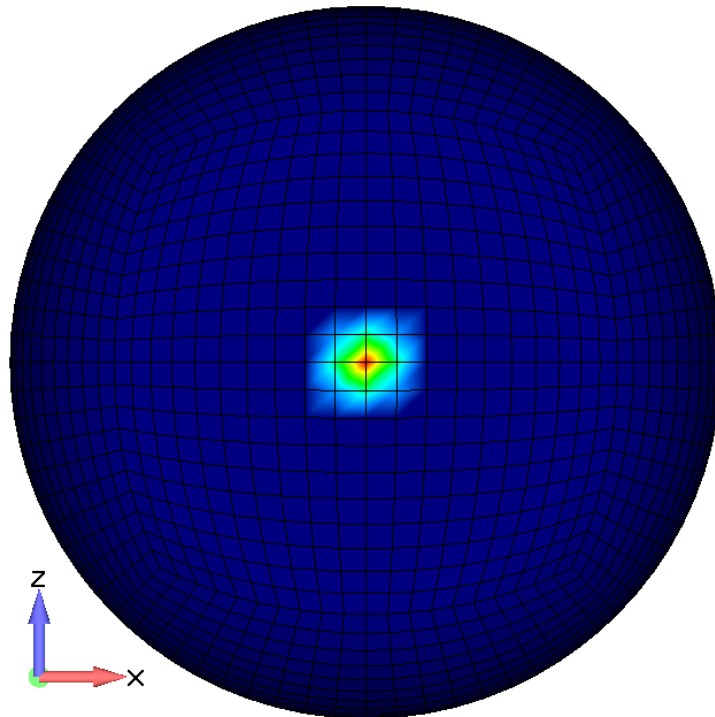
Output Set: Coarsely Meshed Source, NXN 7.0 Refinement
Contour: Contact Pressure
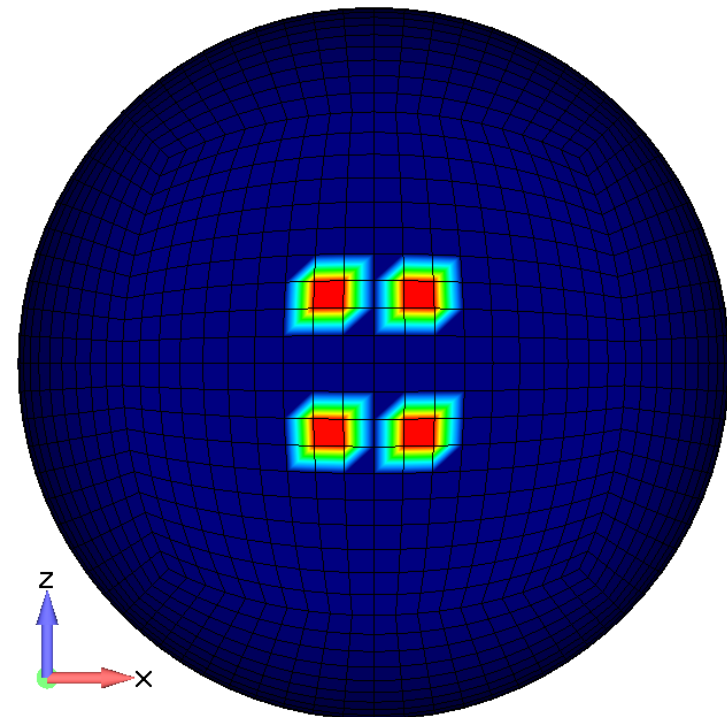
Output Set: Coarsely Meshed Source, No Refinement
Contour: Contact Pressure

Penalty Factor Units - Creates the PENTYP field on the BCTPARM or BGPARM entry. Specifies how contact element stiffness is calculated.

When setting penalty factors for linear contact or glued contact when Glue Type = 1..Spring (GLUETYPE=1)

> 1..1/Length (Default) - Normal Penalty Factor (PENN) and Tangential Penalty Factor (PENT) are entered in units of 1/Length.

> 2..Force/(Length x Area) - Normal Penalty Factor (PENN) and Tangential Penalty Factor (PENT) are entered in units of Force/(Length x Area).

Auto Penalty Factor - This is a flag to indicate whether normal and tangential penalty factors will be automatically calculated. When this box is checked in FEMAP, no special field will be written to the BCTPARM. This is the default for NX Nastran.

Normal Factor - Creates the PENN field on the BCTPARM or BGPARM entry. Designates the penalty factor for the normal direction (Default = 10.0 for BCTPARM, 100 for BGPARM).

Tangential Factor - Creates the PENT field on the BCTPARM or BGPARM entry. Designates the penalty factor for the tangential direction (Default = 1.0 for BCTPARM, 100 for BGPARM).

4.   DATA SURFACE CONNECTION MANAGER

When working with many Connection Regions and Connection Properties the 'Data Surface Editor' may prove helpful. It can be accessed by selecting Tools > Data Surface Editor. Select the appropriate options and click Ok.
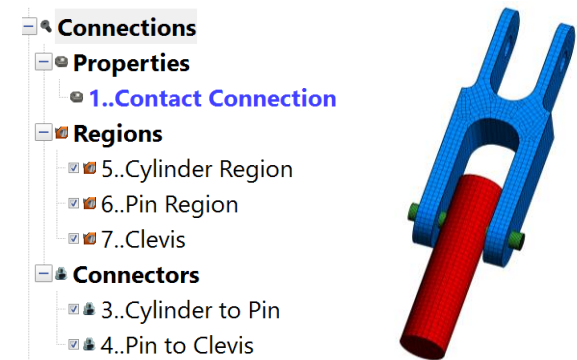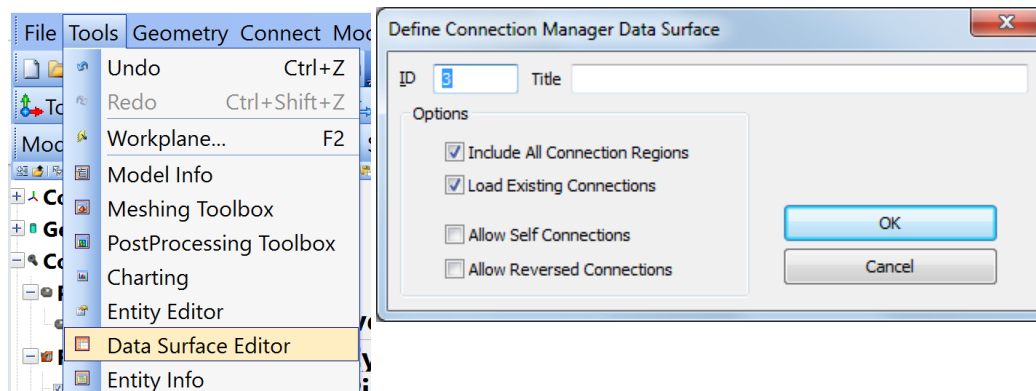
In the Data Surface editor, click "Create/Load Data Surface" dropdown menu and then select "Connection Manager Data Surface". Each Region created will be listed in a column and row, and the user can select a Connection Property in the dropdown menu to create a Connector between the Regions.

The Data Surface Editor on the right displays the same Connectors as the Model Info Tree above.
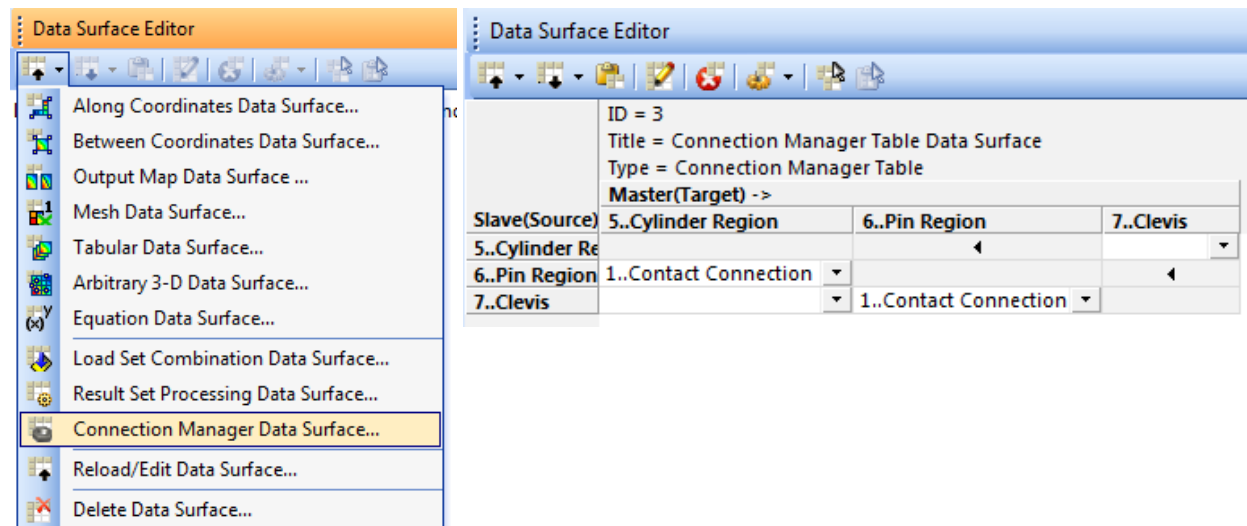
For more information see Femap *Help > Dockable Panes > Data Surface Editor*

5. THERMAL CONTACT FOR GLUED CONNECTIONS

5.1 HAND CALCULATIONS

We are modeling 1D conduction through a 1" x 1" bar having a length of 2.5". A 0.5" section is sliced from the center of the bar and the conduction through that section of bar is replaced with a glued connection. To test the glued connection technology, we want to simulate conduction through a continuous piece of material.

Let's take a look at Fourier's Heat Conduction equation:

$$\frac{q_x}{A} = -k\frac{dT}{dx} \quad or \quad Q = \frac{k}{L}(T_1 - T_2)$$

In our case, we want our glued connection to have the same heat flow and temperature delta as the continuous bar. This means that our heat transfer coefficient is a function of the conductivity and length of the section removed from the bar model.

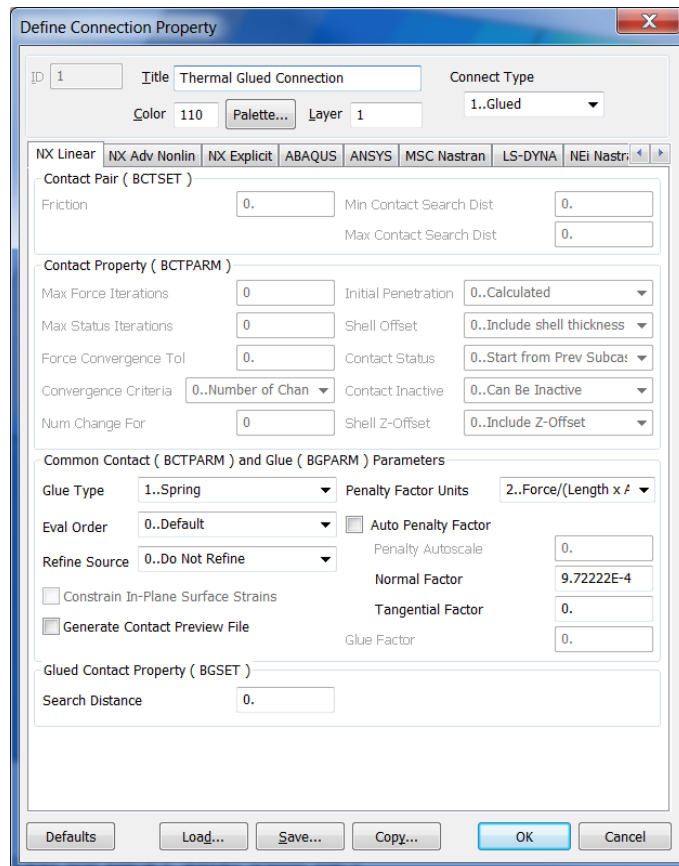$$\frac{Q}{\Delta T} = \frac{4.86e-4\,\dfrac{Btu}{sec*in*°F}}{0.25\,in} = 9.72e-4\,\frac{Btu}{sec*in^2*°F}$$

One could also calculate the heat transfer coefficient between a chip and PCB using the form:

$$\frac{Q}{\Delta T} = \frac{k_{solder}*A_{solder}}{L_{gap}*A_{chip}}$$

5.2 FEMAP AND NX NASTRAN SETUP

To create a glued connection for heat transfer, change the Penalty Factor Units to "2..Force/(Length x Area)". Set the "Normal Factor" equal to your heat transfer coefficient. See NX Nastran User Guide, Section 20.3 *Glue Control Parameters* for more information.

FEMAP Glue Setup	NX Nastran Results

Output Set: NX NASTRAN SS HEAT ANALYSIS SET
Nodal Contour: Temperature