Small Connection Elements (RBE2, RBE3 and CBUSH) and How Amazingly Useful They Are For FEA Modeling

Engineering Mechanics White Paper

FINITE ELEMENT ANALYSIS
Predictive Engineering

Predictive Engineering, Inc.
2505 SE 11th, Suite 310
Portland Oregon 97202
PH: 503.206.5571 FX: 866.215.1220
www.predictiveengineering.com

TABLE OF CONTENTS

1. Summary 6
1.1 SUMMARY OF CONNECTION TECHNOLOGY USING SMALL ELEMENTS 6
1.2 ANALYST RECOMMENDATIONS 6
2. Introduction 7
3. The basics about Multi-Point Constraints (RBE2 and RBE3 Elements) 7
3.1 R-ELEMENTS AND WHY THEY ARE NOT EXACTLY ELEMENTS 7
3.2 SUMMARY OF RBE2 AND RBE3 CHARACTERISTICS 12
3.3 APPLICATIONS USING RBE2 AND RBE3 ELEMENTS 12
3.3.1 THE LOGICAL RBE2 12
3.3.2 THE ILLOGICAL RBE3 14
3.3.3 FORCE SMEARING WITH RBE2 AND RBE3 CONNECTIONS 17
3.3.4 THERMAL (CTE) USAGE OF R-ELEMENTS 19
3.4 SUMMARY OF APPLICATION NOTE ON MPC OR R-ELEMENTS OR RBE2 / RBE3 21
4. Working with the Cbush element as a connection tool 21
4.1 INTRODUCTION 21
4.1 SUMMARY OF CBUSH CHARACTERISTICS 24
4.2 APPLICATIONS USING THE CBUSH ELEMENT 25
4.2.1 BASIC COMPARISONS WITH RBE2 ELEMENTS 25
4.2.2 DIFFICULTIES WITH THE CBUSH ELEMENT 29
4.2.3 SPECIAL APPLICATION OF THE CBUSH ELEMENT 34

All Rights Reserved 2013

4.3 SUMMARY OF CBUSH CHARACTERISTICS 36
5. Appendix 36
5.1 THE UM OPTION FOR THE RBE3 36
5.1.1 WHEN YOU DON'T WANT TO USE THE RBE3 UM OPTION 38
5.1.2 WHAT ABOUT THE CBUSH? 39

List of Figures

Figure 1: How Femap creates a MPC and some terminology from the NX Nastran User Guide. 8
Figure 2: Basic FEA terminology for the way MPC's are handled. 9
Figure 3: A dependent DOF of the RBE2 (node 2) is given a SPC in the Z-direction (circled on the graphic above). The resulting error message calls out the problem using the NX Nastran terminology of "GRID POINT 2 COMPONENT 3 ILLEGALLY…." 10
Figure 4: A six DOF RBE2 was used in Figure 3. A functional RBE3 is shown on the right. Note the Independent DOF's required for stability in the RBE3 formulation. If one does not check the RX box for the RBE3 element, NX Nastran will provide an error message. 11
Figure 5: A column is pulled with a force of 1,000 lbf. A RBE3 is used to uniformly distribute the load. RBE2's are used to simulate a bolted connection to the base structure. 13
Figure 6: A full-on simple example of how a RBE2 works with all six DOF enabled and then reduced to just one DOF. 14
Figure 7: The connection R-element is switched from a RBE2 to a RBE3. Only translation DOF's are used for the independent DOF's on the RBE3. If all six DOF are enforced (left), the result is reasonable whereas with only three translations DOF, the result is not plausible for a bolted connection. 15
Figure 8: With the RBE3, it is impossible to force a rotational stiffness into the structure. As per the formulation, only the load is transferred via the MPC regardless on how the dependent node is defined. Please note how the independent nodes are defined for stability. 16

All Rights Reserved 2013

Figure 9: When transferring load (or weight via a mass element), there is nothing like a RBE3 to do a nice job. Note that the spatial relationship of the force is captured by the R-element (i.e., RZ moment = 20). 18

Figure 10: Result of using a rigid RBE2 for thermal analysis. Note: The only applied load is that of a uniform temperature. 19

Figure 11: How to setup the RBE2 to handle thermal loading. 20

Figure 12: The CBUSH element setup within Femap v11. The column labeled Structural refers to Structural Damping. 22

Figure 13: If the user does not select an Orientation CSys (highlighted), then the X, Y and Z directions align with the DOF's as 1, 2 and 3. The rotational DOF follow this same convention. 24

Figure 14: Femap setup for the CBUSH element. Even though the CBUSH property card was set to use Cartesian CSys, the element definition still requires an Element Orientation Vector (viz., just like a beam element)..... 26

Figure 15: Comparison between RBE2 and CBUSH connection modeling. It is best said that there is no free lunch since the CBUSH element does not quite provide "rigid" type results, unless one uses high stiffness values, which could then lead to numerical problems. 27

Figure 16: Another comparison between a RBE2 and CBUSH connection element. The CBUSH element does not create a rigid mechanism. 28

Figure 17: The obvious setup for a CBUSH element attached to a solid element where the load is vertical (Y-direction)..... 29

Figure 18: The CBUSH nodes are coincident and the load case is a 100 lbf vector load in 1,1,1 direction. The CBUSH element is stable. 30

Figure 19: The solid element only has translational DOF and if the CBUSH tries to attach to nothing, NX Nastran will respond appropriately. 31

Figure 20: With the end node of the CBUSH constrained in RX, RY and RZ, the element is stable. 32

Figure 21: With a stable arrangement, the CBUSH can transfer moment loads onto solid elements; however, the stiffness of the CBUSH element is likewise transferred unlike the RBE3. 33

Figure 22: The CBUSH elements with rotational constraints coupled with the RBE2 elements in a cylindrical coordinate system provides a smooth torsion transition along the tube. 35

All Rights Reserved 2013

Figure 23: An internal pressure is applied to the cylinder. The RBE3 provides constraint without rigidity. The dependent node is SPC'ed in all six DOF at the center. This is impossible without transferring these SPC's to the U_m DOF. 37

Figure 24: Setting up the RBE3 with the Um option to provide restraint to the internally pressurized cylinder. 38

Figure 25: Another approach to solving the internal pressurized cylinder with an RBE2 connection. 39

Figure 26: Using the CBUSH Element to obtain the same result as shown for the RBE3 and RBE2 connections. 40

All Rights Reserved 2013

1. SUMMARY

1.1 SUMMARY OF CONNECTION TECHNOLOGY USING SMALL ELEMENTS

This white paper assumes that the reader has the basics of FEA down pat and an inkling of how R-elements work. The objective is to describe in detail how to use R-connections and CBUSH elements correctly and with confidence. If you make it through this note, you'll most likely know more about these little connections than 99% of your peers.

We'll cover the basics of MPC terminology which is the foundation of the RBE2 and RBE3 connections. A keen understanding will be provided on how to think in terms of independent and dependent nodes. It'll be obvious after this discussion that it is not logical to apply SPC's to dependent nodes or to connect other dependent nodes between different R-elements. Best practices will be covered and some recommendations given.

The thermal CTE capability of the RBE2 connection will also be covered for completeness.

Lastly, the CBUSH element will be introduced and applications given on how to use this replacement for the CELAS element. The downfalls on using this element will be discussed and also why this element is a useful as a companion to the RBE2 and RBE3 elements.

1.2 ANALYST RECOMMENDATIONS

- ✓ R-elements (RBE's) are not elements but multi-point-constraints (MPC's) that just happen to look like elements graphically.
- ✓ Never apply SPC's too dependent nodes.
- ✓ Try not to daisy chain RBE's.
- ✓ R-elements work at the constraint level and offer a powerful idealization tool that ensures that the stiffness matrix will not become ill-conditioned as is possible with spring elements.
- ✓ R-elements are linear. If the analysis is expected to be nonlinear use Sol 601 where the R-element formulation is switched automatically to a spring-type formulation.
- ✓ Thermal CTE capability is possible with R-elements. The element is switched from Rigid to Lagrange.
- ✓ CBUSH elements are the go-to-replacement for the CELAS element.
- ✓ CBUSH is the multi-function, six DOF spring element that correctly models the spring behavior between non-coincident nodes.

All Rights Reserved 2013

2. INTRODUCTION

This white paper will walk you through the theory underlying multi-point constraints (RBE2 & RBE3), CBUSH, CGAP and CBEAM elements and how to use these elements to create efficient and elegant FEA models. Our goal is to provide a working knowledge of this technology and we encourage you to build your own FE models to verify the concepts presented in this paper. As we tell many of our support clients, there is nothing like structured organic learning.

3. THE BASICS ABOUT MULTI-POINT CONSTRAINTS (RBE2 AND RBE3 ELEMENTS)

3.1 R-ELEMENTS AND WHY THEY ARE NOT EXACTLY ELEMENTS

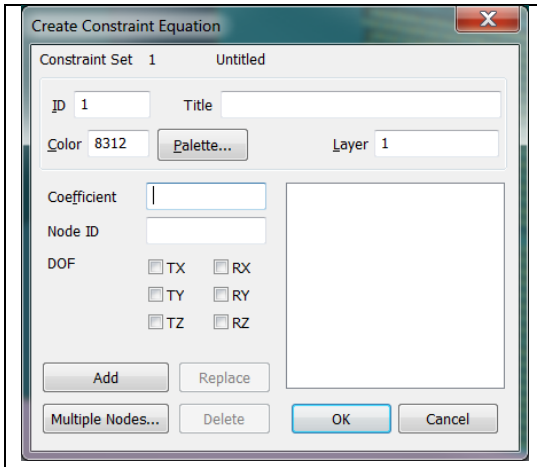
The tomes that have been written about R-type elements number in the dozens. In their usage they have friends and foes. For one highly experienced Nastran engineer that I know, the use of RBE3 elements is generally prohibited among their crew simply due to their ability to suck the life (i.e., load) out of a standard analysis run and such death-inducing ability is often reason enough to just say "no" to their usage.

To get a handle on how R-elements work, it is best to think of them as multi-point constraints (MPC). Which then leads to the question as to what exactly is a MPC? Taking a look at the NX Nastran Element Library Reference, one is presented with some dense logic but at its core, it defines a MPC as a linear equation among chosen displacements that is equal to zero. Here's the mathematical form of a MPC:

$$A_1 \cdot u_1 + A_2 \cdot u_2 + A_3 \cdot u_3 + A_4 \cdot u_4 + A_5 \cdot u_5 + A_6 \cdot u_6 + A_7 \cdot u_7 + \text{etc.} = 0$$

An MPC creates a linear relationship ("**A**'s") between the user specified degrees-of-freedom ("**u**'s"). In Femap, the user can create MPC's directly or indirectly with the use of R-elements. In the latter case, NX Nastran pre-defines the constants ("A's") to match the required behavior. When the analysis deck (*.DAT) file is submitted to the NX Nastran solver, the MPC's are treated as a special form of a constraint. For those of you that have had graduate courses in FE or have attended one of our Femap and NX Nastran classes, a light bulb should be going off about now. The word constraint says it all and a further definition is given in Figure 1. In the analysis sequence, the stiffness matrix (K) is formed and then constraints are applied to this matrix. At this stage, any user defined SPC's are applied and the MPC relationship is enforced between the required **u**'s. In the MPC process (or say R-element process), dependent degrees-of-freedom (DOF) are made slaves to the independent DOF. A little matrix math is performed and the dependent DOF's are removed from consideration within the displacement DOF matrix set. This should make sense in the terminology of independent versus dependent. If a DOF is dependent then it is toast and one can't apply a SPC to this DOF. This is a pretty crucial bit of information and the source of many technical support phone calls. Before this point is further elaborated, take a look again at Figure 1 then we'll continue.

All Rights Reserved 2013



9.1 Introduction to Constraints

A constraint is the enforcement of a prescribed displacement (i.e., component of translation or rotation) on a grid point or points. There are two basic types of constraints in NX Nastran: single point constraints (SPCs) and multipoint constraints (MPCs).

- A single point constraint is a constraint applied to an individual grid point. Single point constraints can enforce either zero displacement or nonzero displacement.
- A multipoint constraint is a mathematical constraint relationship between one grid point and another grid point (or set of grid points).

The boundary conditions of a static structure (fixed, hinged, roller support, etc.) typically require that various degrees of freedom be constrained to zero displacement. For example, consider a grid point fixed in a rigid wall. All six displacement degrees of freedom—three translational directions and three rotational directions—must be constrained to zero to mathematically describe the fixed boundary condition.

Real world structures often don't have simple or ideal boundary conditions. Because a model's constraints greatly influences its response to loading, you must try to constrain your model as accurately as possible.

This chapter describes how you apply constraints. To understand how constraints are processed, you need to be familiar with the NX Nastran set notation and matrix operations.

Figure 1: How Femap creates a MPC and some terminology from the NX Nastran User Guide.

It is nigh near impossible to reinforce this concept of independent / dependent DOF's without a little matrix math. Along with this matrix math, we'll introduce the jargon that NX Nastran uses for error messages when the user applies a SPC to an R-element dependent DOF. Figure 2 shows a sequence of equations that should help to illustrate the MPC relationship to the FEA process.

$$F = [K]\{u_i\}$$

All Rights Reserved 2013

The MPC (or R-element):

$$[A_i]\{u_i\} = 0$$

Is then defined as independent / dependent DOF's:

$$\{u_i\} = \begin{Bmatrix} u_s \\ u_m \end{Bmatrix}$$

Where u_s is the set of independent DOF's and u_m is the set of dependent DOF's.

Using these relationships, the dependent DOF's are defined as:

$$\{u_m\} = [A_i^*]\{u_s\}$$

Where A^* is a special transform of the MPC constants (see NX Nastran theory manual).

Figure 2: Basic FEA terminology for the way MPC's are handled.

With the dependent DOF's removed, it seemed a bit obvious that NX Nastran would bark at any user that had applied a SPC to any dependent DOF of an R-element. An example of such an error message is given in Figure 3.

All Rights Reserved 2013

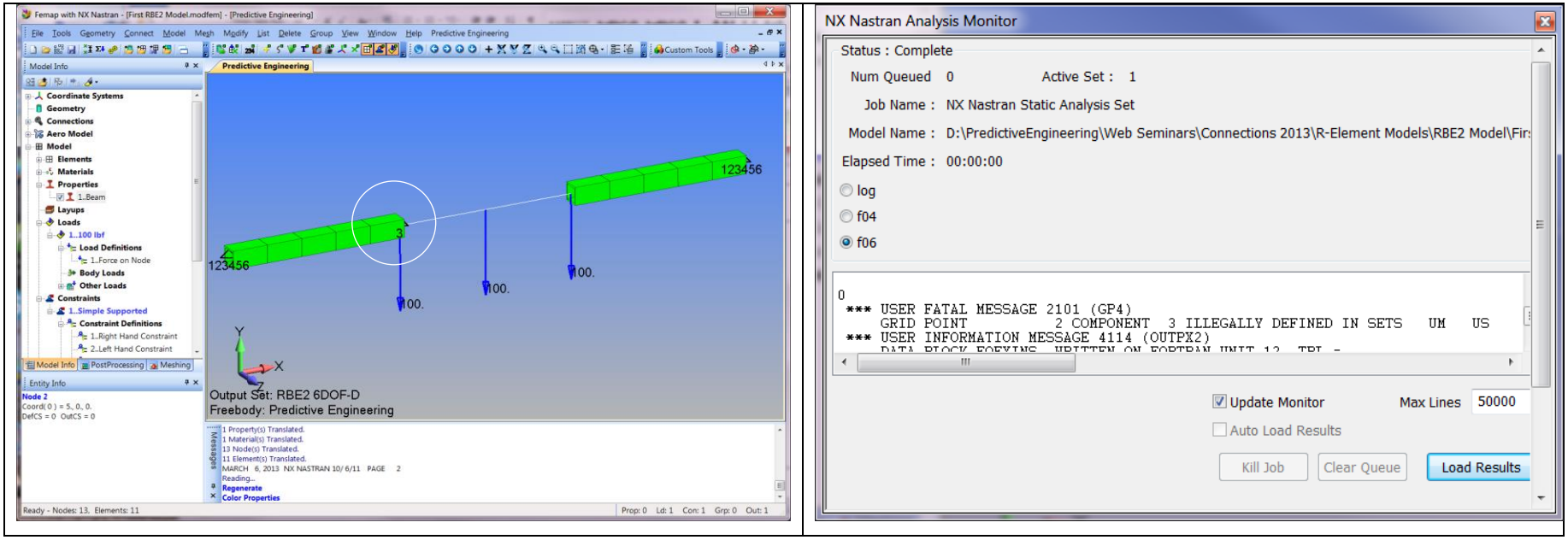


Figure 3: A dependent DOF of the RBE2 (node 2) is given a SPC in the Z-direction (circled on the graphic above). The resulting error message calls out the problem using the NX Nastran terminology of "GRID POINT 2 COMPONENT 3 ILLEGALLY…."

One may note that the model happily accepts the forces applied to the two dependent DOF's of the RBE2. This is logical given that forces have nothing to do with the {u} column matrix and one can apply forces to either independent and/or dependent DOF's.

Although we haven't formally defined the RBE2 and RBE3 element, one would most likely not be reading this white paper unless you had some inkling of their function. But just to be formal, here's a brief description:

- RBE2: A MPC that defaults with six DOF for its independent node and the user can then choose what DOF to enforce upon the dependent nodes. The MPC creates a rigid mechanism between the independent and dependent nodes.
- RBE3: A MPC that defines the constraint behavior of the independent nodes as a function of its dependent nodes. The element has no default DOF and the user must choose the MPC's constraints carefully to prevent a free body mechanism. The MPC interpolates the overall constraint behavior of the dependent nodes upon the independent node. This formulation imparts no rigidly between

All Rights Reserved 2013

the independent and dependent nodes. One key application is the smearing of a force from the independent node to the dependent nodes.

Now, let's take the given RBE2 above and morph it into a RBE3. Figure 4 shows the setup for this transformation. Since RBE3's are meant for interpolating between the dependent / independent nodes, NX Nastran gives you the option to scale this dependency with a Factor. Within NX Nastran they call this a local weight factor and the default Factor is 1. This implies that the dependent node is equally influenced by the independent nodes. If one is really clever and has done their reading, in some rare cases, it makes sense to change this factor but honestly I have not stumbled across any good reasons. The other odd looking option is UM DOF in the dependent node section. This is a rather rare and unique option but it boils down to letting the user apply a SPC to the dependent DOF on the RBE3 element. Although this is typically a no/no it does this trick by taking nodes from the independent set and trying them back into the MPC to allow the dependent node to be constrained. If this topic is of interest, take a look at the Appendix.

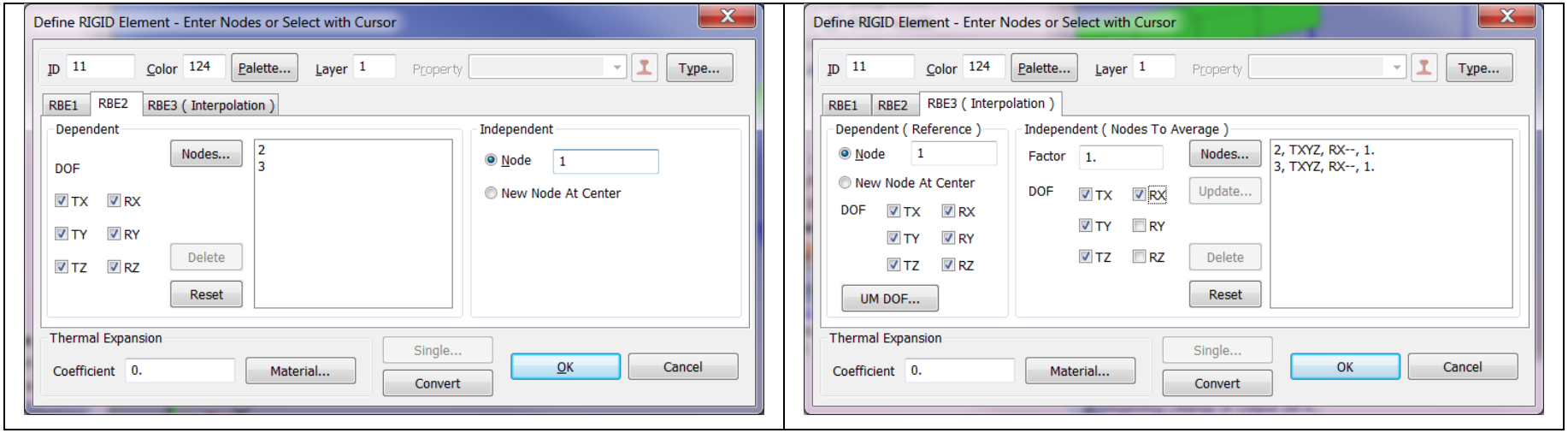


Figure 4: A six DOF RBE2 was used in Figure 3. A functional RBE3 is shown on the right. Note the Independent DOF's required for stability in the RBE3 formulation. If one does not check the RX box for the RBE3 element, NX Nastran will provide an error message.

All Rights Reserved 2013

In the exact formulation shown in Figure 4 the model solves with no problem. As with the RBE2 element, applying forces to either the dependent/independent DOF's is allowable. If one has been paying attention to this discussion, one may ask what about applying SPC's to RBE3 dependent DOF? Well, that is a question that is best left to the inquisitive mind by building and running a small RBE3 model; don't want to spoon feed all the good stuff.

3.2 Summary of RBE2 and RBE3 Characteristics

- At heart they are MPC's.
- If one grasps the concept of independent and independent DOF's in their usage, every common problem can be avoided.
- With the above in mind, here are three rules for using RBE elements or R-elements safely:
    - Never have a dependent node defined in another R-element.
    - Never have a dependent node defined in another R-element.
    - Never have a dependent node defined in another R-element.
- RBE3 elements can be brutal. Think of them as small little free bodies floating in space. They need to have sufficient DOF defined to be stable but no more. An over constrained RBE3 is illogical and NX Nastran will do unexpected illogical things to the solution. A couple of rule-of-thumbs are provided:
    - Only use translation DOF for the independent DOF. If rotation DOF's are absolutely necessary, test it as if your life depended upon it.
    - Do not hook a RBE3 into a RBE2 unless you really know what you are doing.
    - RBE3's are awesome elements and do a great job of distributing load without enforcing rigidly. They can be extremely useful if used with a bit of understanding.

3.3 Applications using RBE2 and RBE3 Elements

3.3.1 The Logical RBE2

The usage of the RBE2 is fairly trivial for someone experience in the ways of NX Nastran. That is, if you don't apply SPC's to the dependent DOF's or tie in other RBE2's or RBE3's, you'll have smooth sailing. The key concept to a RBE2 is that it is a rigid mechanism that is defined

All Rights Reserved 2013

via a linear relationship among its selected nodal DOF. Since it is defined as a constraint relationship, it is formulated once at the start of the solution and then used henceforth forward. If a nonlinear solution is performed (i.e., SOL 106), don't expect a RBE2 (likewise a RBE3) to know the difference between a linear and nonlinear deflection. It is a little caveat that can sneak up and bite you if you are running a lot of SOL 106 analyses. As an analyst's note, if you run the NX Advanced Nonlinear Solution (SOL 601), it will convert your RBE2/RBE3 elements into another non-MPC formulation.

To start out this application section, a joint connection is modeled using a beam element and two RBE2 elements. Figure 5 shows the results for this configuration with the RBE2 set to constrain the dependent nodes at all six DOF's and then just the translation DOF's.

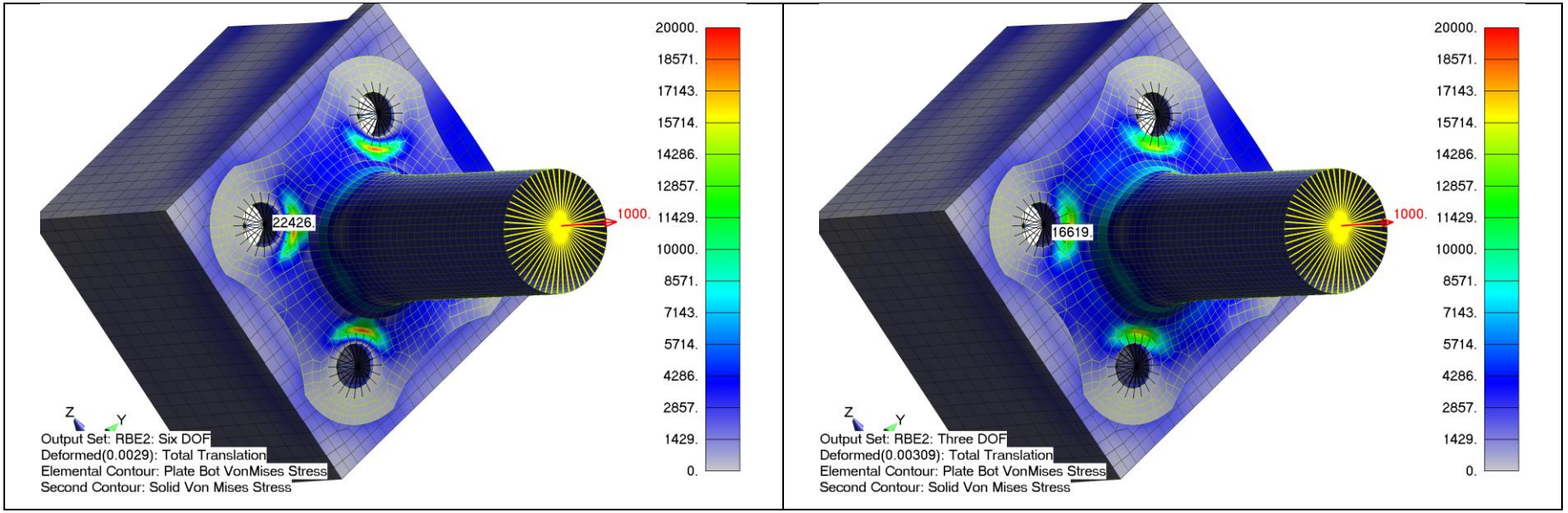


Figure 5: A column is pulled with a force of 1,000 lbf. A RBE3 is used to uniformly distribute the load. RBE2's are used to simulate a bolted connection to the base structure.

All Rights Reserved 2013

What I like about this example model shown in Figure 5 is that both solutions look realistic. The only difference is how rigid one makes the RBE2 by the use of additional DOF applied to the *dependent* DOF's. In this example it makes sense that as one completely constrains the dependent DOF (left) the stresses would be higher than that on the less constrained RBE2 (right).

A good homework exercise for RBE2 elements is to create a small model and then play around with the dependent DOF's of the MPC. A prior seminar was done on connections in 2011 (see Predictive Engineering Resources web page under Seminars) where the model shown in Figure 6 was used to explore this concept.

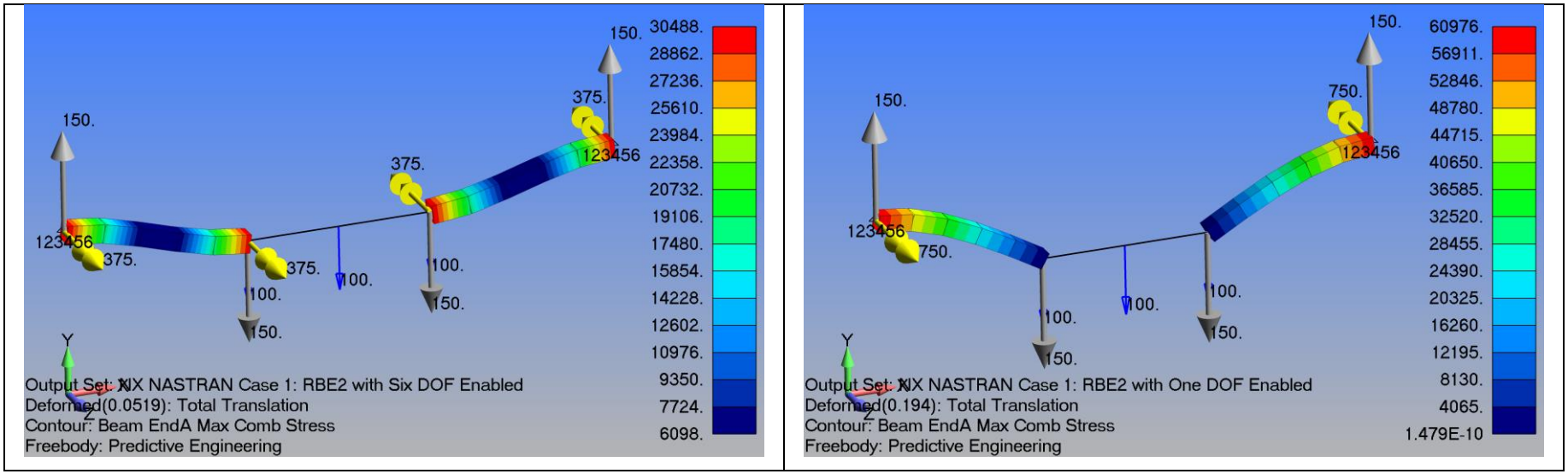


Figure 6: A full-on simple example of how a RBE2 works with all six DOF enabled and then reduced to just one DOF.

At this stage, the R-element model shown in Figure 6 should be self-evident. One of the nice things about this figure is the use of the free-body-diagram (FBD) to verify or illuminate the behavior of this MPC (remember that RBE's are actually not elements).

3.3.2 THE ILLOGICAL RBE3

The next example shown in Figure 7 is that for the RBE3 where the same switch is used between the dependent node. As given in our guidance, only translational DOF are used for the independent nodes thus leaving only the dependent node to play around with. The

All Rights Reserved 2013

results shown in Figure 7 are stark showing the huge differences that can happen with an interpolation element that does not impart stiffness but only attempts to balance the relative displacement movement between the independent to dependent nodes.

As in the prior example with RBE2 connections, a simple example is presented in Figure 8 to help explain how the MPC nicknamed “RBE3” works. The difference between these R-element formulations is immediately apparent since the RBE3 cannot provide any stiffness between the independent and dependent nodes, the beam structure is not stiffened and only forces are transferred. This is an extremely useful feature for the modeling of many features where the analyst does not want to impart stiffness into the structure.

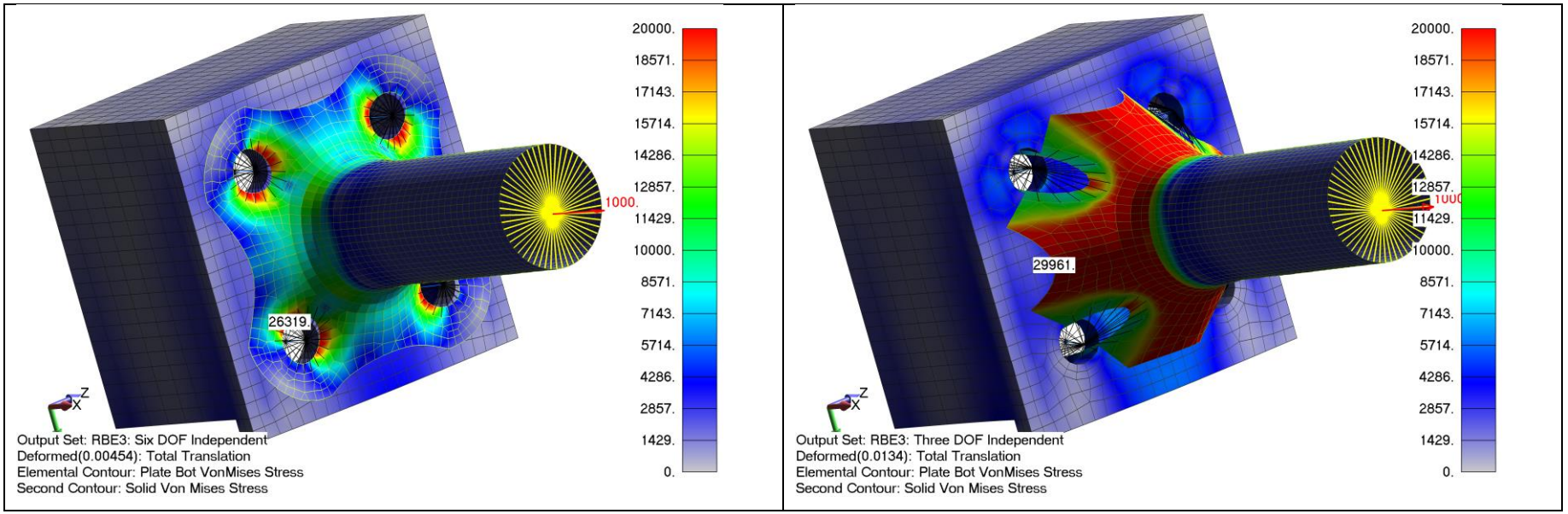


Figure 7: The connection R-element is switched from a RBE2 to a RBE3. Only translation DOF’s are used for the independent DOF’s on the RBE3. If all six DOF are enforced (left), the result is reasonable whereas with only three translations DOF, the result is not plausible for a bolted connection.

All Rights Reserved 2013

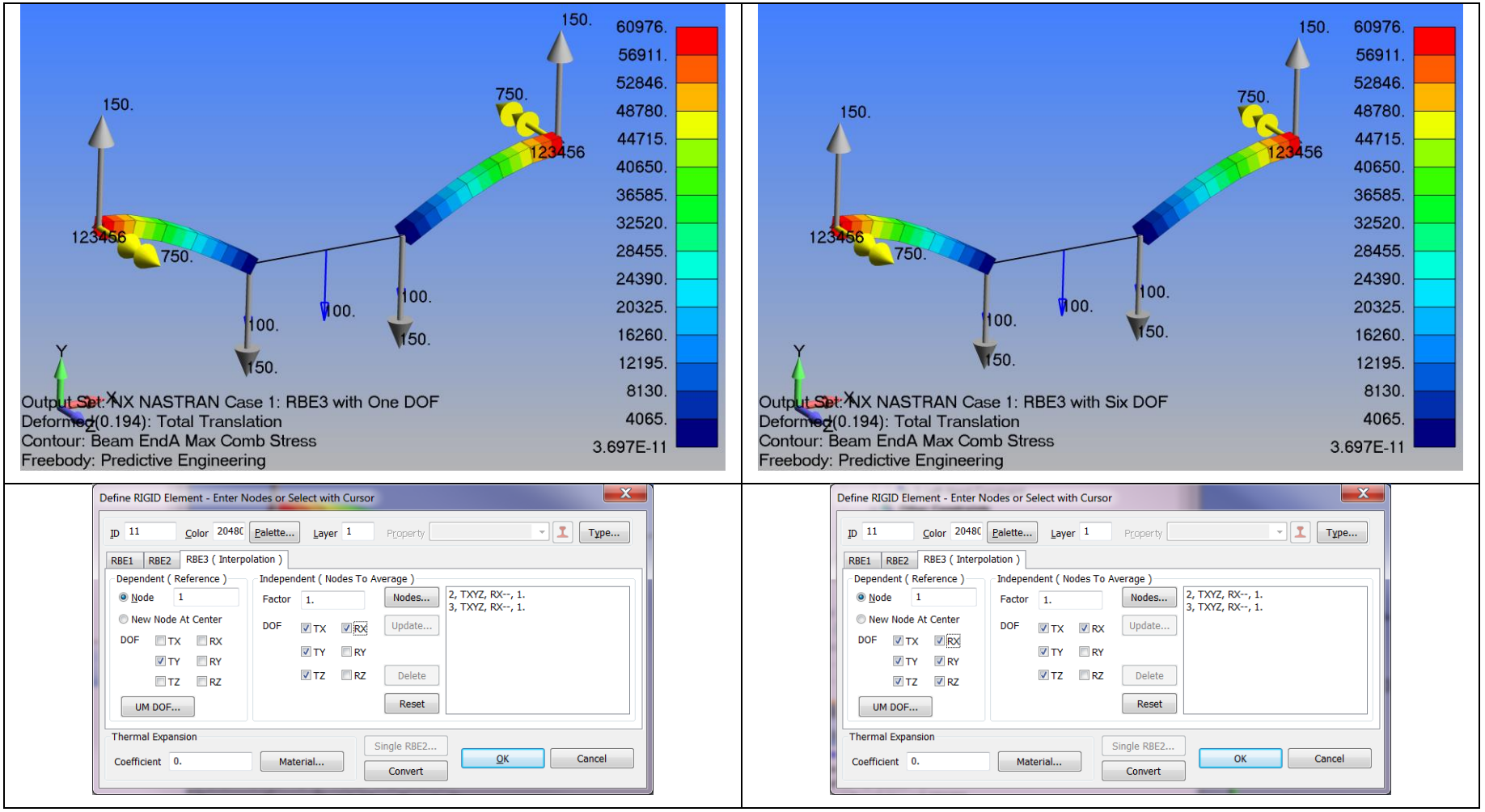


Figure 8: With the RBE3, it is impossible to force a rotational stiffness into the structure. As per the formulation, only the load is transferred via the MPC regardless on how the dependent node is defined. Please note how the independent nodes are defined for stability.

All Rights Reserved 2013

3.3.3 FORCE SMEARING WITH RBE2 AND RBE3 CONNECTIONS

The last example that is shown in Figure 9 is that for a mass element attached onto some brick elements. The model is loaded with a body load and the objective is to transfer the weight of the given object smoothly onto the structure with no artificially induced stiffness. If one tries to do this with a RBE2, one induces a rigid mechanism onto the structure and the force transfer could be considered artificial. When the R-element is switched over to a RBE3 element, the stress results show a nice smooth pattern.

To verify the behavior of the R-element, FBD's were included. The mass element with a weight of 10 lbf was placed at 2" from the center of the tube. Given the 1 g loading, it is heartening to see that a RZ moment of 20 lbf-in was calculated for both element types.

One little detail to note is that since we are attaching to solid elements, the respective dependent (RBE2) or independent (RBE3) nodes can only represent translation DOF since solid elements only have translation elements. It is a rather small detail since NX Nastran automatically ignores the user's request for rotation DOF, but if someone else reviews your model and notices such modeling detail they will wonder if you know what they know or just think that you really don't know what you are doing.

All Rights Reserved 2013

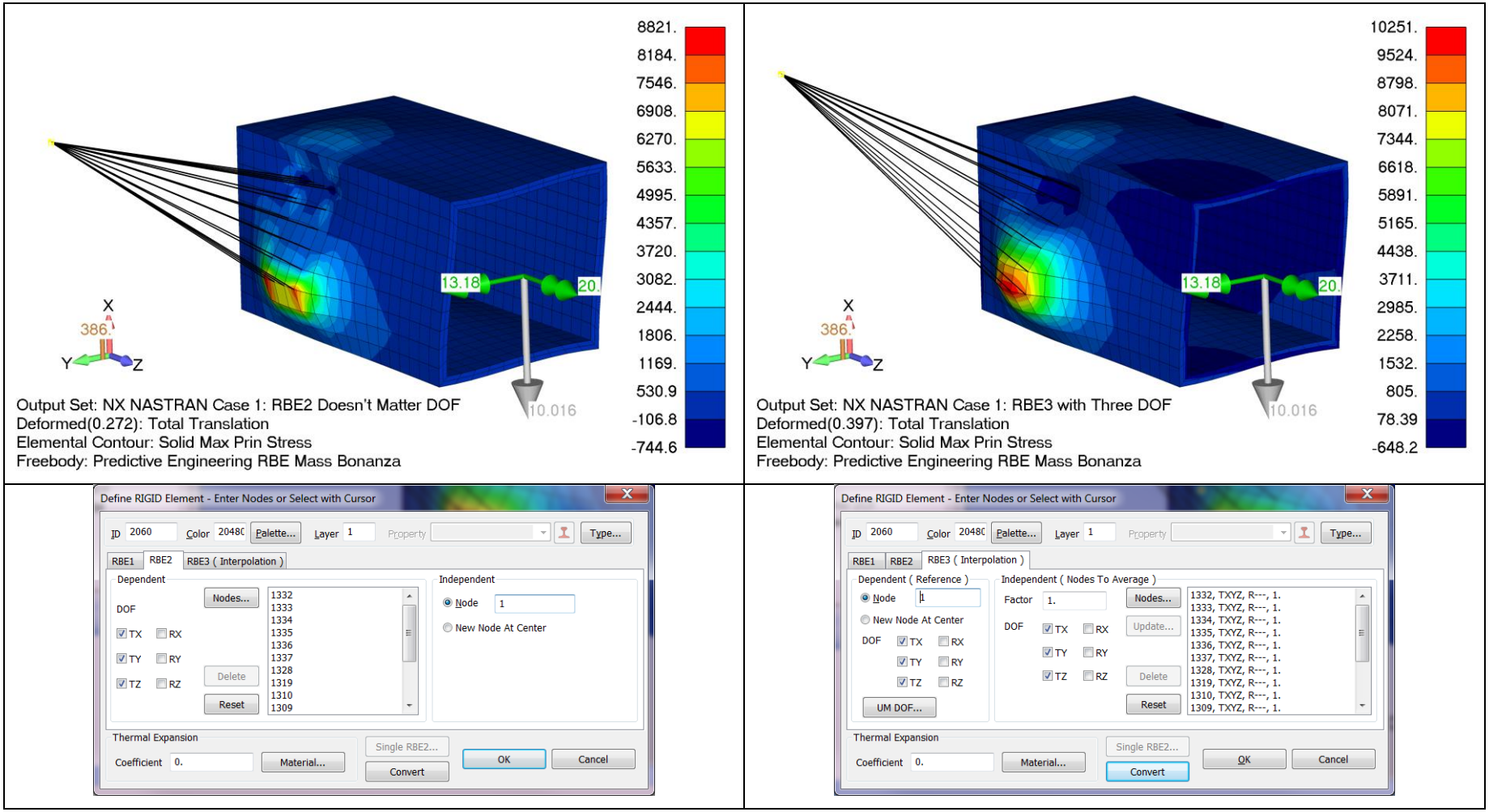


Figure 9: When transferring load (or weight via a mass element), there is nothing like a RBE3 to do a nice job. Note that the spatial relationship of the force is captured by the R-element (i.e., RZ moment = 20).

All Rights Reserved 2013

3.3.4 THERMAL (CTE) USAGE OF R-ELEMENTS

There is another facet to using MPC's that should be mentioned and that has to do with thermal-stress work. A common experience is for analysts to create bolted connections using RBE2 elements and then need to add in the effects of thermal stress. Let's take the prior model from Figure 5 and throw a thermal load onto the top column structure of 100 degrees. We'll use a play CTE value of 1e-5. Figure 10 shows what happens when thermal considerations are ignored. Since the rigid link is rigid with no knowledge about a CTE value, it stays rigid and restricts the expansion of the surrounding material.

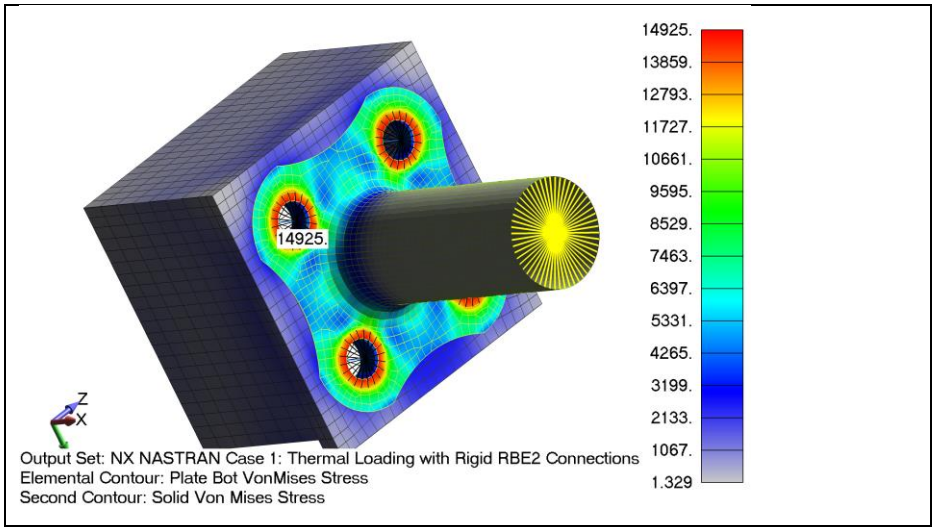


Figure 10: Result of using a rigid RBE2 for thermal analysis. Note: The only applied load is that of a uniform temperature.

Since this is a somewhat common practice, Nastran allows the option of adding a CTE to the RBE2 connection. However, when this is done the element formulation is no longer a MPC but one that adds stiffness terms to the [K] matrix. The formulation switch is called LAGRAN in Nastran'ese since it uses a technique called "Lagrange Multiplier Method". The short story is that Nastran handles this switch efficiently and rarely does the Lagrange multiplier method produce an ill-conditioned stiffness matrix. Figure 11 shows the results for this switch. One RBE2 element is left in its standard formulation (which is denoted as LINEAR within the NX Nastran Element Library Reference).

All Rights Reserved 2013

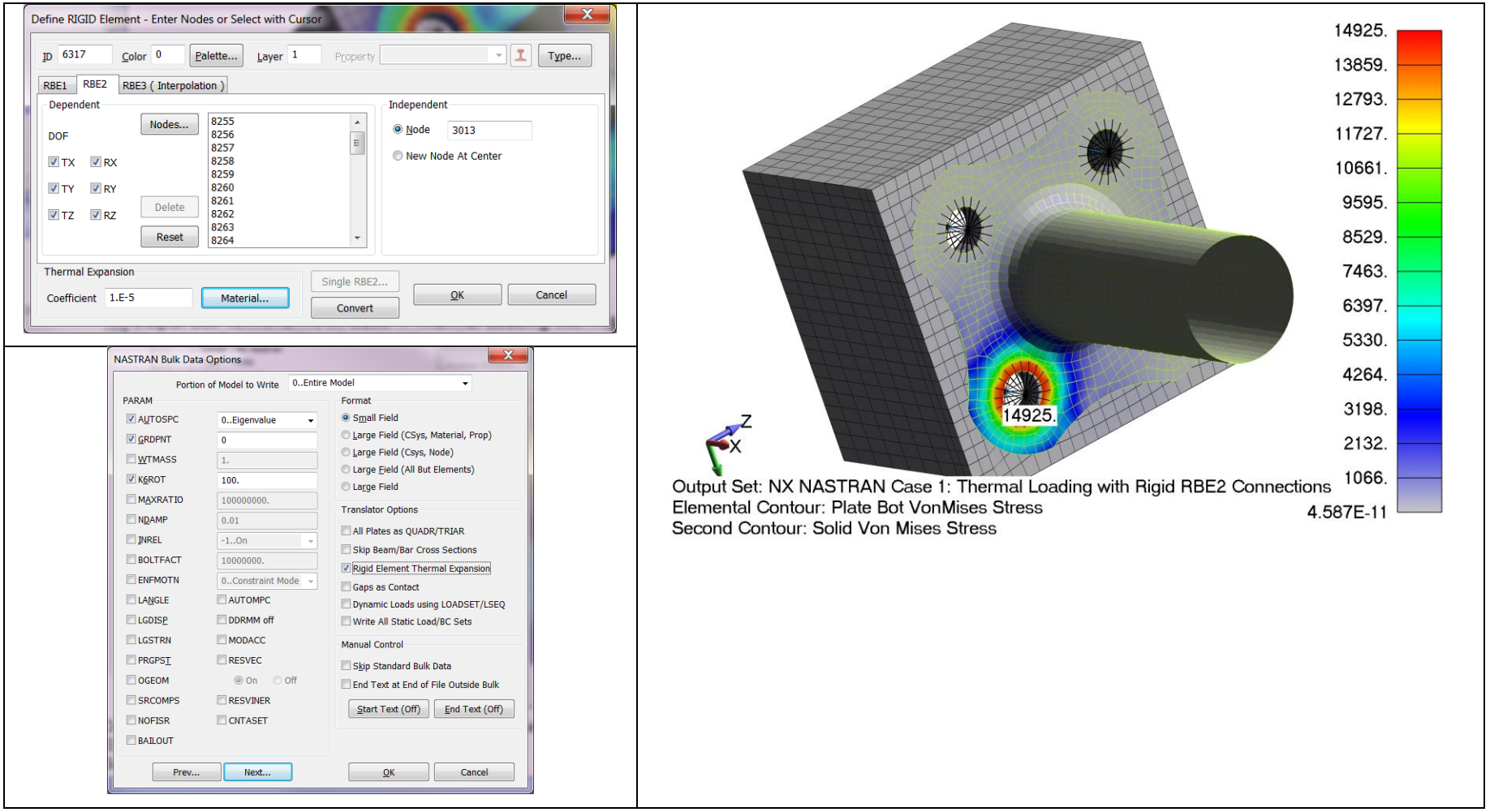


Figure 11: How to setup the RBE2 to handle thermal loading.

All Rights Reserved 2013

3.4 SUMMARY OF APPLICATION NOTE ON MPC OR R-ELEMENTS OR RBE2 / RBE3

Here are the highlights from this section:

- RBE2 transfer stiffness as a rigid body among the independent/dependent node set.
- The analyst can control the DOF's where stiffness is transfer but at the end of the day – stiffness is added.
- A RBE2 is always numerically stable (no free body mechanisms) since the independent node starts out with a full set of DOF's.
- RBE2 and RBE3 elements both correctly capture the spatial relationships between the independent/dependent nodes.
- RBE3 elements can create free body mechanisms if the selected nodes do not create a stable "mechanism".
- The force interpolation capability of the RBE3 element is often worth the hassle of dealing with its special restrictions.
- MPC's operate at the constraint level and do not create stiffness terms within the [K] matrix. Although this makes these connections only suitable for analysis work that is strongly linear, they also do not perturb the matrix with high stiffness terms as spring elements might do to obtain equivalent behavior.
- If thermal CTE is desired be aware that this switches the R-element to a Lagrange formulation and stiffness terms are added to the [K]. Although this method is efficient, the R-element is no longer a MPC and it is possible to ill-condition the stiffness matrix [K].

4. WORKING WITH THE CBUSH ELEMENT AS A CONNECTION TOOL

4.1 INTRODUCTION

The CBUSH element is now the default spring element for Femap users as shown in Figure 12.

All Rights Reserved 2013

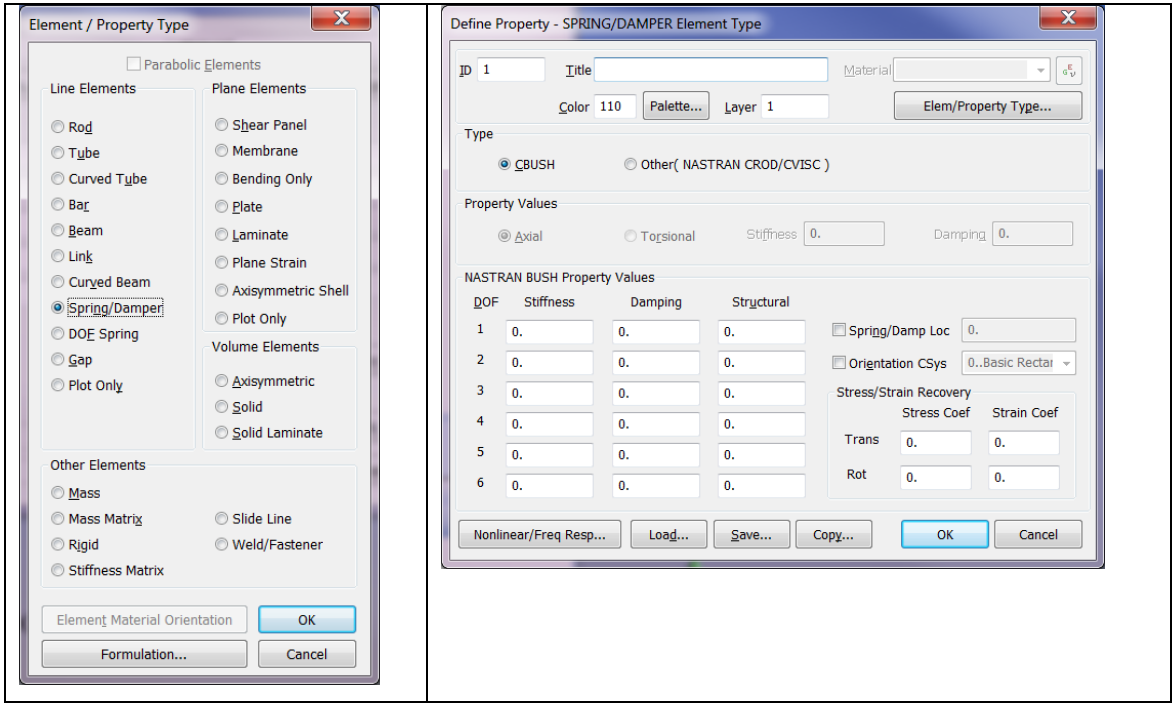


Figure 12: The CBUSH element setup within Femap v11. The column labeled Structural refers to Structural Damping.

Historically and numerically there are problems noted with the use of the CELAS formulations when the two nodes that define the spring element are not coincident. This is pointed out in the NX Nastran User's Guide:

> ## Zero-dimensional Elements
> When you use CELASi elements to represent concentrated springs between two components of translation, the directions of the two components must be coaxial. Even small deviations in direction can induce a significant moment to your model that does not exist in your physical structure. When you use a CELASi element, the locations of the two end points should be coincident to avoid this type of problem. If the two end points aren't coincident, consider using a CROD or CBUSH element instead.

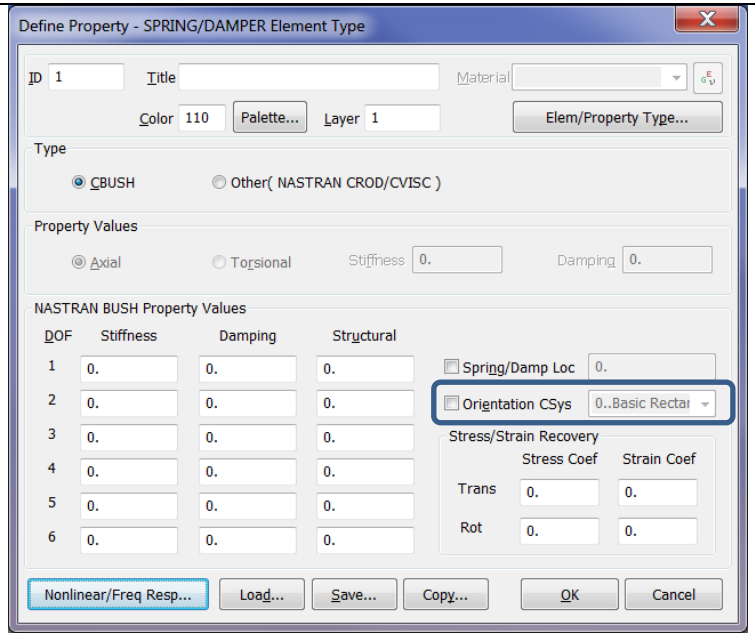
All Rights Reserved 2013

Given this uncertainty and a preference for not having coincident nodes within a model where merge operations might occur (although Femap's smart merge capability would never merge two nodes connected with an element), we have standardized on only using the CBUSH for our spring'ey needs. For anecdotal evidence, this principal engineer once used a nested mess of CELAS elements on a large model and had this exact same effect occur. The sad part was that this error was pointed out to me by the client's chief engineer. I felt pretty small. A rather complicated example of how to screw up your model with CELAS elements is given in the NX Nastran Element Library if you feel so compulsive.

The CBUSH element should be treated as you would treat a beam element with six DOF's. Once this adjustment has been made, their rational use is not such a big deal. Figure 13 shows a schematic from the NX Nastran Quick Reference Guide. As with beam elements, the user needs to define the orientation vector of the element to fix the Y-direction (DOF 2) of the element. For this reason, our standard practice, if possible, is to define the CBUSH using the standard global Cartesian CSys and avoid this bit of modeling.

Another consideration about CBUSH elements is that they can be used in all solution sequences and, in particular, they are correctly translated for SOL 601 (NX Advanced Nonlinear) to handle large deformations. As one may remember, MPC's (i.e., R-elements) are defined within the constraint matrix and are strictly for linear mechanics. This is an advantage of the CBUSH element is its ability to morph correctly for the SOL 601 analysis. If the user is interested in running the standard NX Nastran nonlinear analysis (SOL 106) and large deflections are foreseen, then NX Nastran has the CBUSH1D formulation. I would rather not dwell on this odd-ball formulation since if you really need large deflection; one is better served with the CBUSH element and SOL 601. If this topic peaks your interest, take a look at the NX Nastran Basic Nonlinear Analysis Users Guide and the Advanced Nonlinear Theory and Modeling Guide.

All Rights Reserved 2013



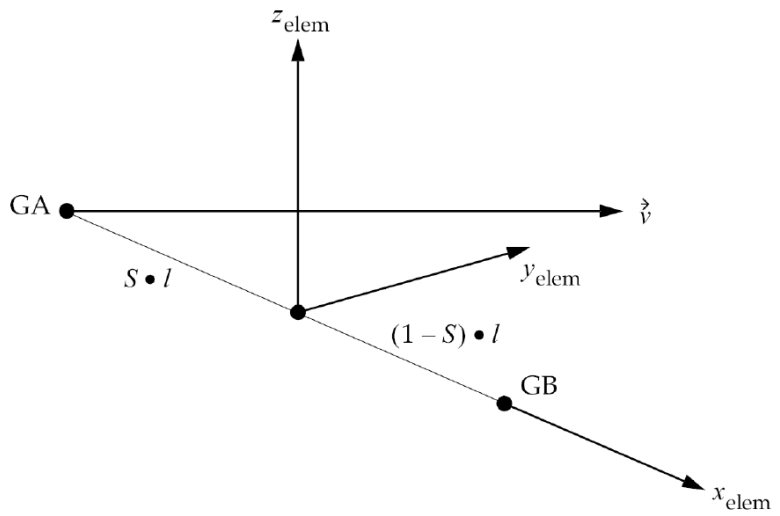


Figure 12-16. CBUSH Element

Figure 13: If the user does not select an Orientation CSys (highlighted), then the X, Y and Z directions align with the DOF's as 1, 2 and 3. The rotational DOF follow this same convention.

4.1 SUMMARY OF CBUSH CHARACTERISTICS

- The CBUSH formulation is stable and proven with no surprises. It is the go forward spring element for all seasons.
- It is the infinitely variable stiffness beam element. The user can control any DOF.
- Nonlinear capability built into the element along with advanced dynamic features.
- The only downside is that it has six DOF's and the user has to make some mental adjustments when using these elements.
- The user should remember that they are adding stiffness terms into the stiffness matrix [K] and that if one choose's large CBUSH values it could cause unexpected numerical difficulties. Note: MPC's are based on constraints and do not affect the [K].

All Rights Reserved 2013

4.2 Applications using the CBUSH Element

4.2.1 Basic Comparisons with RBE2 Elements

To not get too far ahead of ourselves, Figure 14 shows how the CBUSH element is defined within Femap. Even though we had set the CBUSH property card to use a global Cartesian CSys, it still requires the user to define an Element Orientation Vector. The first example will start out with mimicking the behavior of RBE2 element shown in Figure 6 by removing the current RBE2 and replacing it with a very stiff CBUSH element as visually described in Figure 15. If one sets the stiffness the 6th DOF within the CBUSH card, the resulting reaction moments (RZ) are somewhat like the prior RBE2 setup. The difference is that the element is free to pivot at the center whereas the RBE2 is a "rigid mechanism". This result is given in Figure 16.

All Rights Reserved 2013

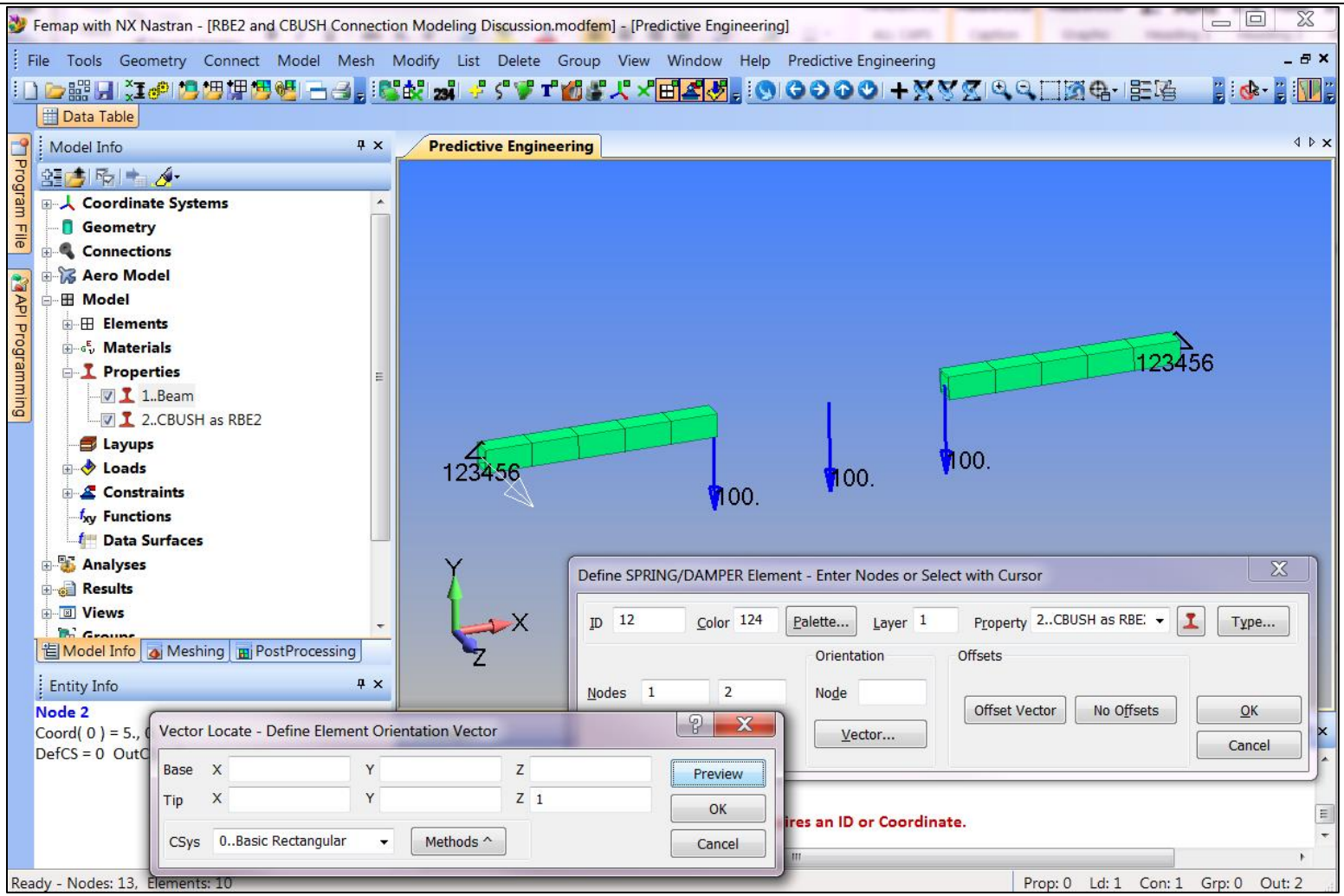


Figure 14: Femap setup for the CBUSH element. Even though the CBUSH property card was set to use Cartesian CSys, the element definition still requires an Element Orientation Vector (viz., just like a beam element).

All Rights Reserved 2013

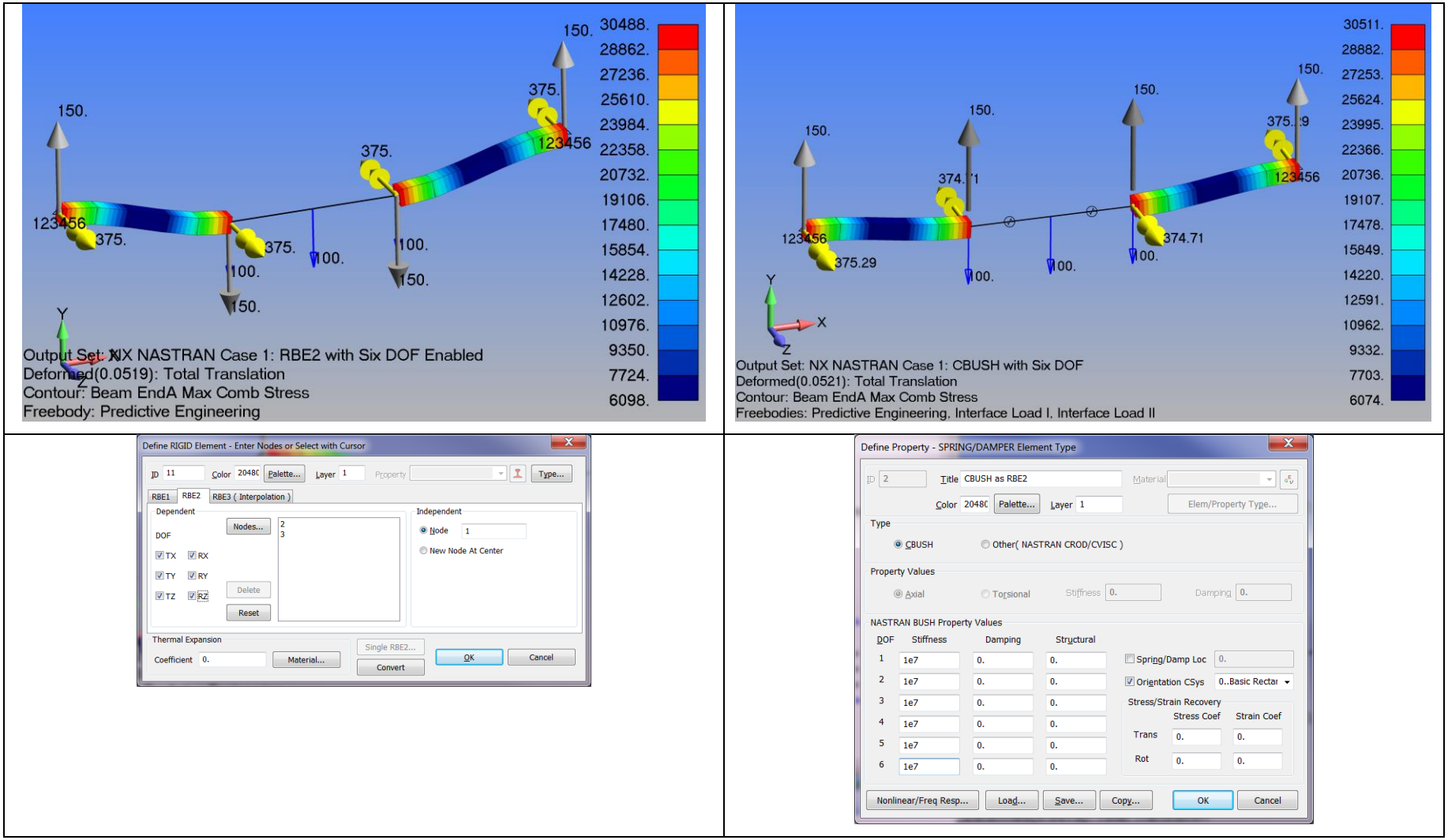


Figure 15: Comparison between RBE2 and CBUSH connection modeling. It is best said that there is no free lunch since the CBUSH element does not quite provide "rigid" type results, unless one uses high stiffness values, which could then lead to numerical problems.

All Rights Reserved 2013

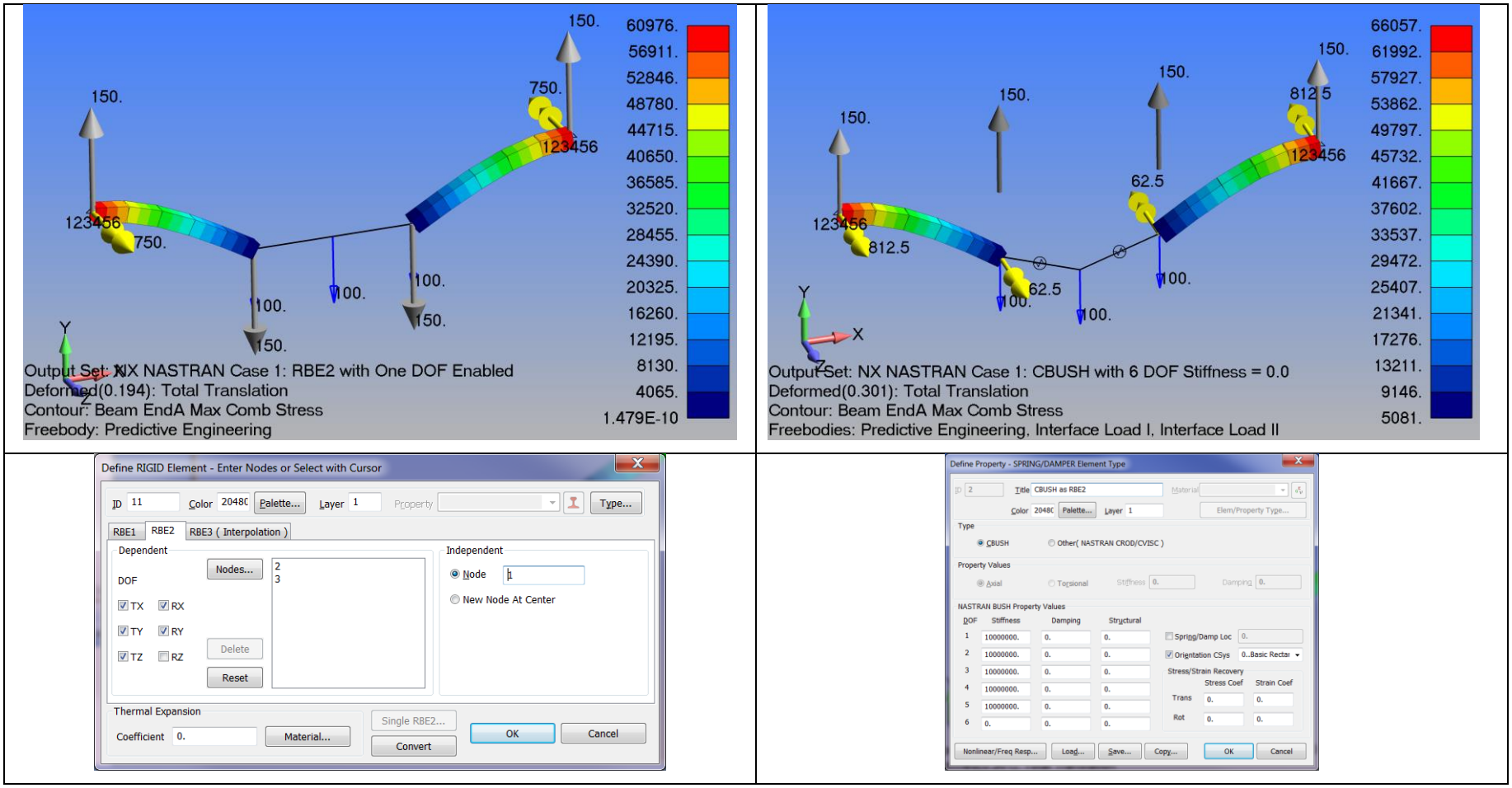


Figure 16: Another comparison between a RBE2 and CBUSH connection element. The CBUSH element does not create a rigid mechanism.

All Rights Reserved 2013

4.2.2 DIFFICULTIES WITH THE CBUSH ELEMENT

The biggest stumbling block with the CBUSH element is that it requires a little thinking in its application. To illustrate this point, let's take a solid element and pull on it with a CBUSH element. The CBUSH is vertically aligned with the top corner node of a brick element. The load is then aligned to pull in this direction.

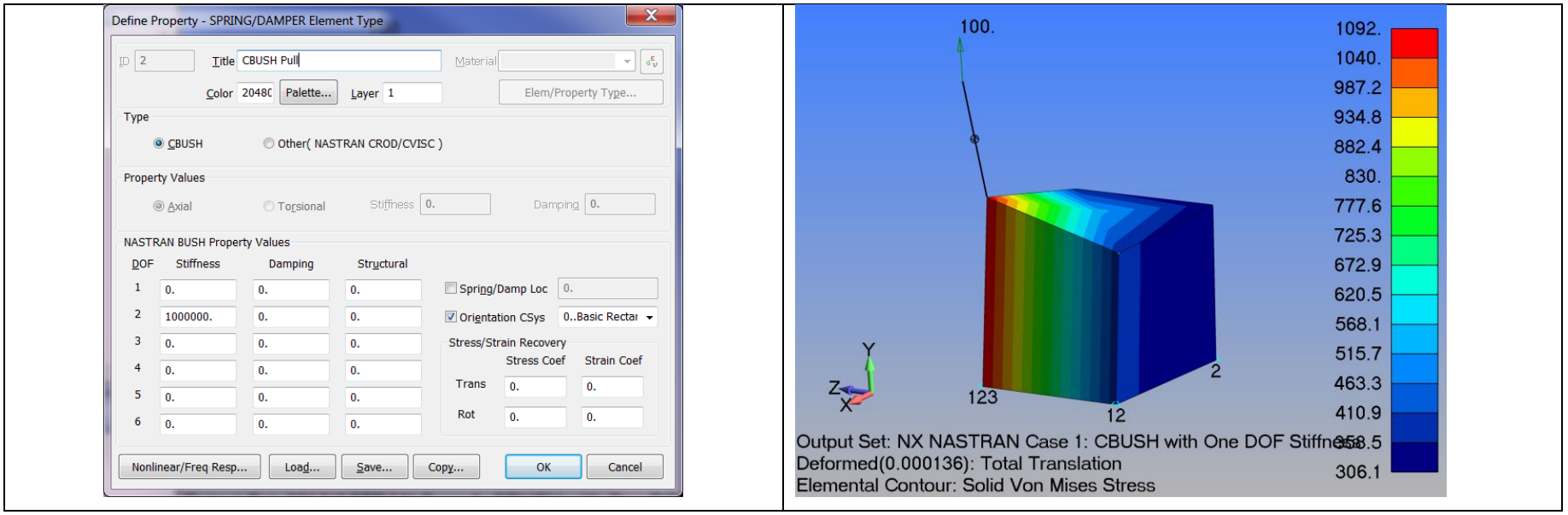


Figure 17: The obvious setup for a CBUSH element attached to a solid element where the load is vertical (Y-direction).

This is the only configuration that works. If we move the top node of the CBUSH element just the tiniest distance out-of-alignment, the solution fails. For example, we take the top node of the CBUSH and move it 0.001" in the Z-direction, the run will fail with an error message as "RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS IN MATRIX KLL" with the R1 DOF listed as the problem. This makes sense if one works through the DOF logic. That is, the solid element only supports translation DOF and no rotational stiffness is available for the CBUSH element to grab onto and furthermore our CBUSH element is setup to not have any rotational stiffness. With the element offset in the Z-direction, the element is unstable in the RX (R1) DOF. Let's show another example where we make the CBUSH element coincident with the

All Rights Reserved 2013

corner node and apply a force in all three directions. In this example, the CBUSH element will have an initial stiffness in the 1, 2 and 3 directions. Figure 18 shows the results for this test case.

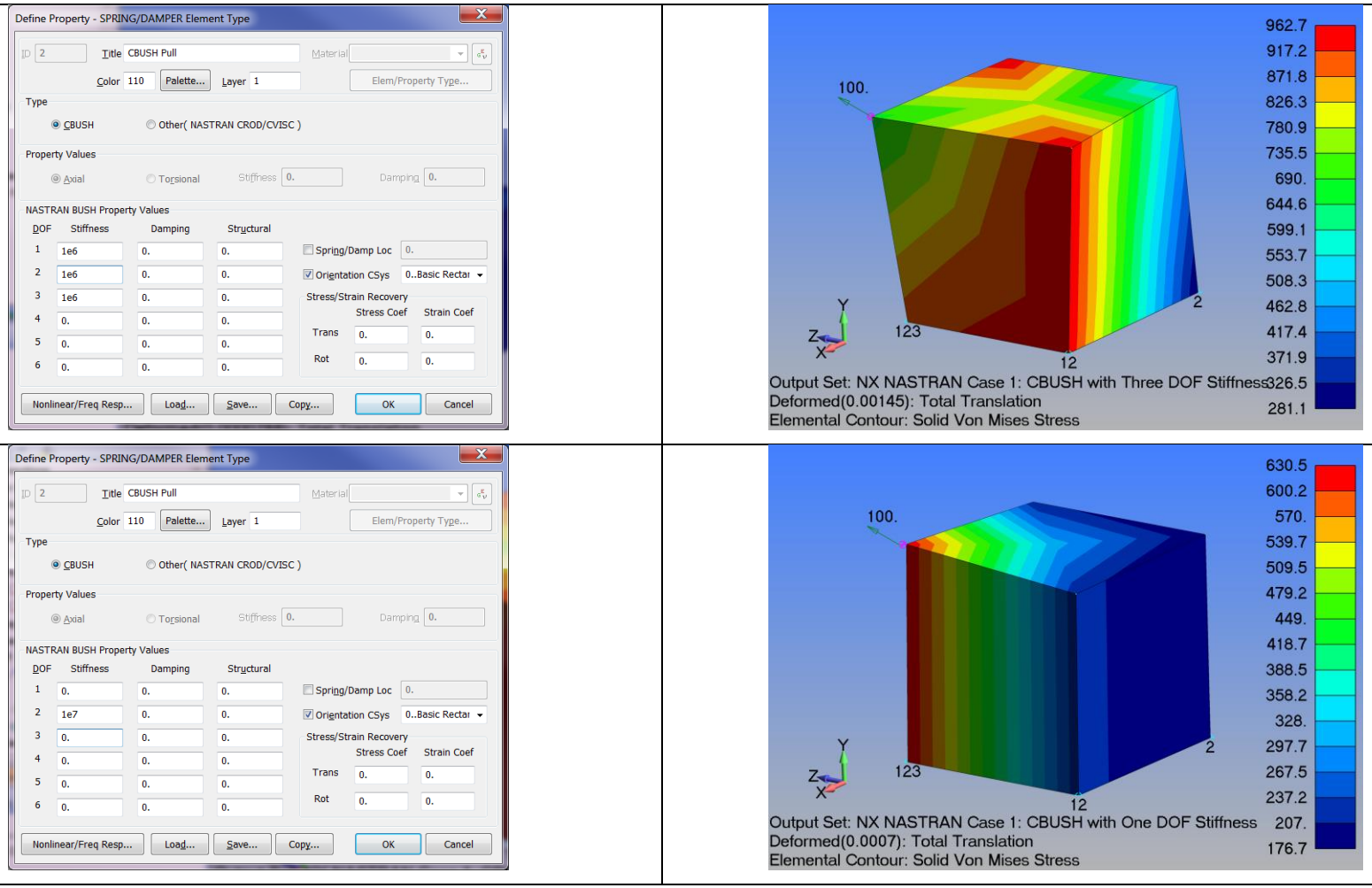


Figure 18: The CBUSH nodes are coincident and the load case is a 100 lbf vector load in 1,1,1 direction. The CBUSH element is stable.

All Rights Reserved 2013

What is interesting and logical, that if the CBUSH nodes are coincident, the element does not contain any rotational free-body-mechanisms. More importantly, this example encourages the reader to focus on how the FEA method works. The first step is the formulation of the stiffness matrix followed by the application of constraints (SPC's and MPC's). The stiffness matrix is then decomposed for the application of forces. If the stiffness matrix is non-singular (i.e., excessive pivot ratios in the matrix KLL), the run will abort. Loads have nothing to do with the analysis running or not running. Given this understanding, it makes sense that if one removes the stiffnesses in the DOF 1 and 3 within the CBUSH element, the solution would proceed as if nothing is unusual. But you're not out of the woods quite yet. If the CBUSH element is defined to have a rotational stiffness, then the solution will fail predictably (see Figure 19).

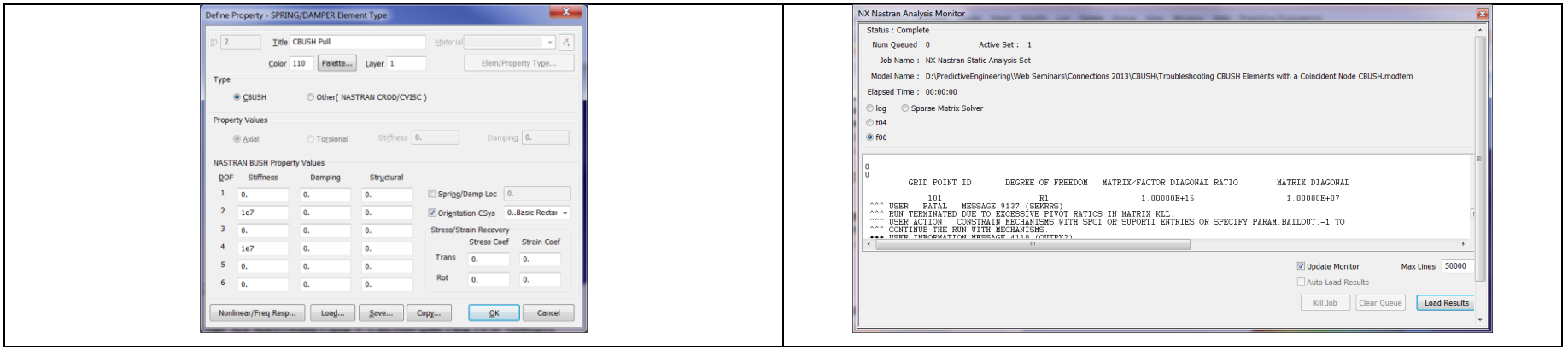


Figure 19: The solid element only has translational DOF and if the CBUSH tries to attach to nothing, NX Nastran will respond appropriately.

For the sake of conversation, let's say that you needed to apply a rotational load through this one CBUSH element? It would be impossible without adding a plate element or some beams to the top surface of the brick element to provide the additional rotational DOF. Saying that, there are some other tricks one can do. Figure 20 shows a setup where the CBUSH element is in the position as shown in Figure 17 (not coincident) but with its end node constrained in all rotational DOF (RX, RY and RZ). Since the CBUSH element has stiffness in the rotation DOF's (4, 5 and 6); the element is stable and NX Nastran solves appropriately.

Another example is that where a group of CBUSH elements can likewise provide stability for the application of a moment load. Figure 21 shows this arrangement and demonstrates that the utility of the CBUSH for a wide range of load application.

All Rights Reserved 2013

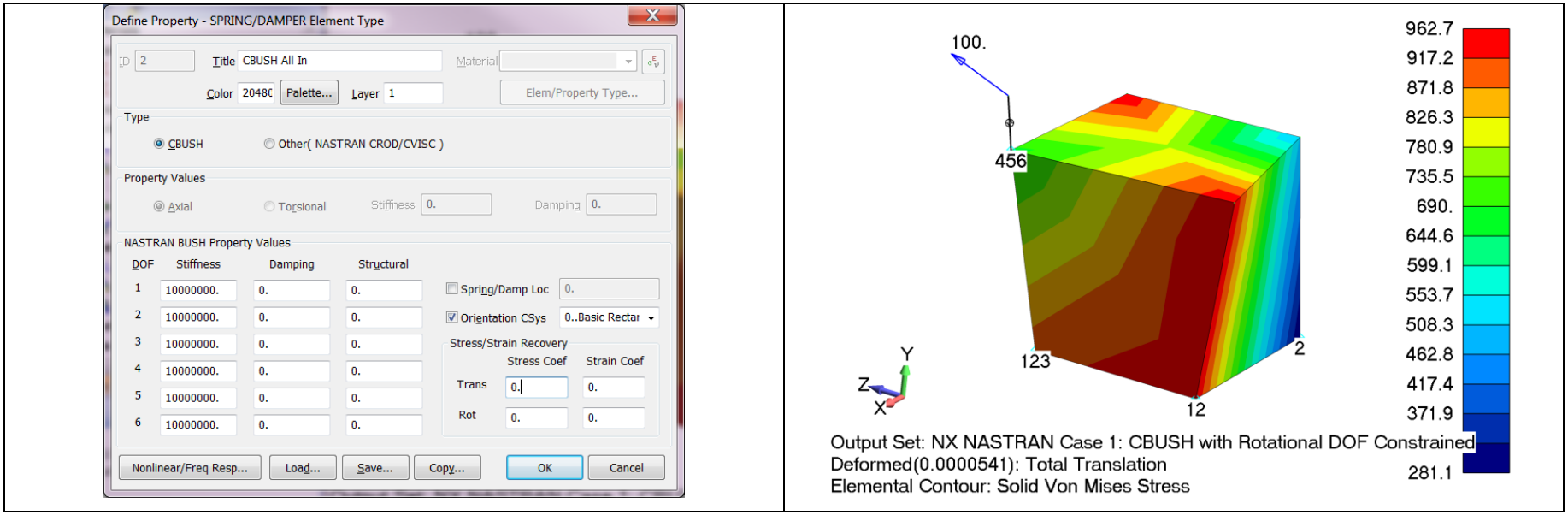


Figure 20: With the end node of the CBUSH constrained in RX, RY and RZ, the element is stable.

All Rights Reserved 2013

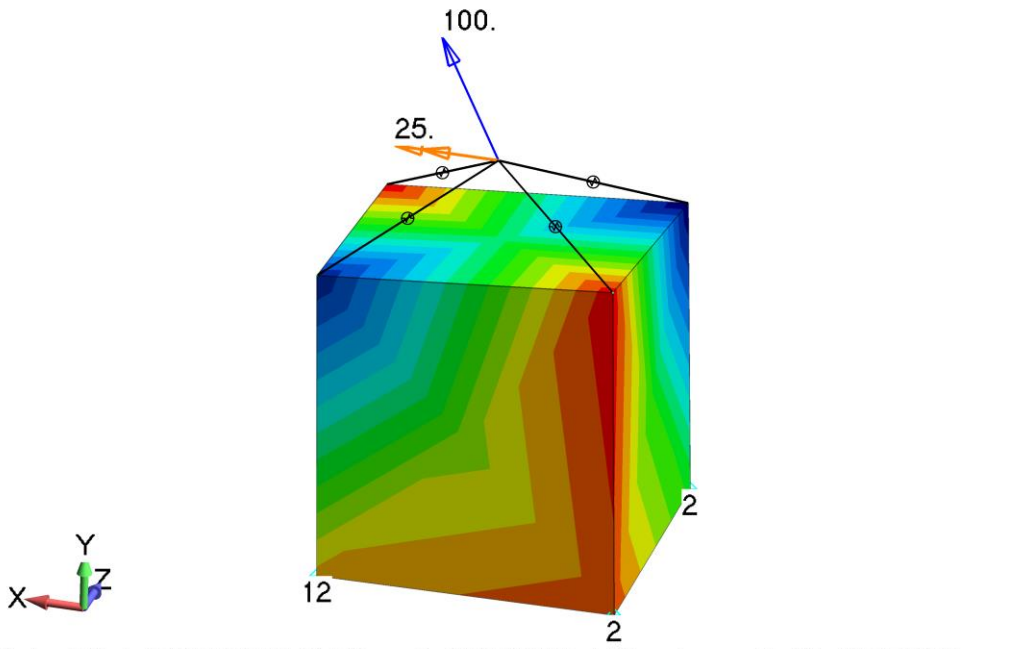


Figure 21: With a stable arrangement, the CBUSH can transfer moment loads onto solid elements; however, the stiffness of the CBUSH element is likewise transferred unlike the RBE3.

All Rights Reserved 2013

4.2.3 Special Application of the CBUSH Element

The last example shows how to use the CBUSH element as a torsion coupler between two hollow rods (see Figure 22). To join the two cylinders that have a slight gap between them, a series of CBUSH elements were created between the two surfaces using Femap's Mesh / Connect / Closest Link command. They are shown graphically as the dense mass of lines and numbers in the middle of two cylinders. As discussed earlier, to allow the CBUSH elements to account for moments, one end of the CBUSH elements had SPC's applied in the 4, 5 and 6 DOF's. If one zooms in on the graphic, these numbers can almost be discerned.

For the torsion load application and to cleanly constrain the tube, RBE2's were used. The procedure was to create a user defined cylindrical coordinate system centered down the axis of the cylinder. The nodes of both ends of the tube were then modified to have their nodal output coordinate system in this cylindrical coordinate system (Femap command Modify / Update Other / Output CSys). In this manner, the RBE2 connection dependent DOF's are in the coordinate system of R, theta and Z.

Given this starting point, the logical question is why the torsion load was applied using a RBE2 and not a RBE3 which has been advertised as the perfect force smearing connection. This opens up a whole can of worms but maybe it is best that we peer into this can all together. The difficulty or challenge with the RBE3 on this particular application is that the RBE3, with the default weighting factor of 1.0, distributes the moment equally to all the attached nodes. For the torsion to be smoothly applied, the quantity of moment (resolved force per node) needs to be scaled per the area fraction of the node. In other words, the shear (force/area) needs to be smooth across the end section. To get this behavior with a RBE3 would require a bit of effort to calculate individual weighting factors for each ring of nodes. Whereas, if one uses the RBE2, it just enforces a fixed displacement in the theta direction and we are good to go. The moral of this example is to not get fixed in your ways but be cognizant of the objective you're trying to achieve.

At the other end, the end constraint using a RBE2 is easily explained since the RBE2 only has two active dependent DOF's in the theta and Z direction. The independent node is SPC'ed in all six DOF and correctly constrains the structure.

Although this example appears a bit cumbersome, it covers a nice twist to the standard plug and chug examples where everything is obvious and transparent.

All Rights Reserved 2013

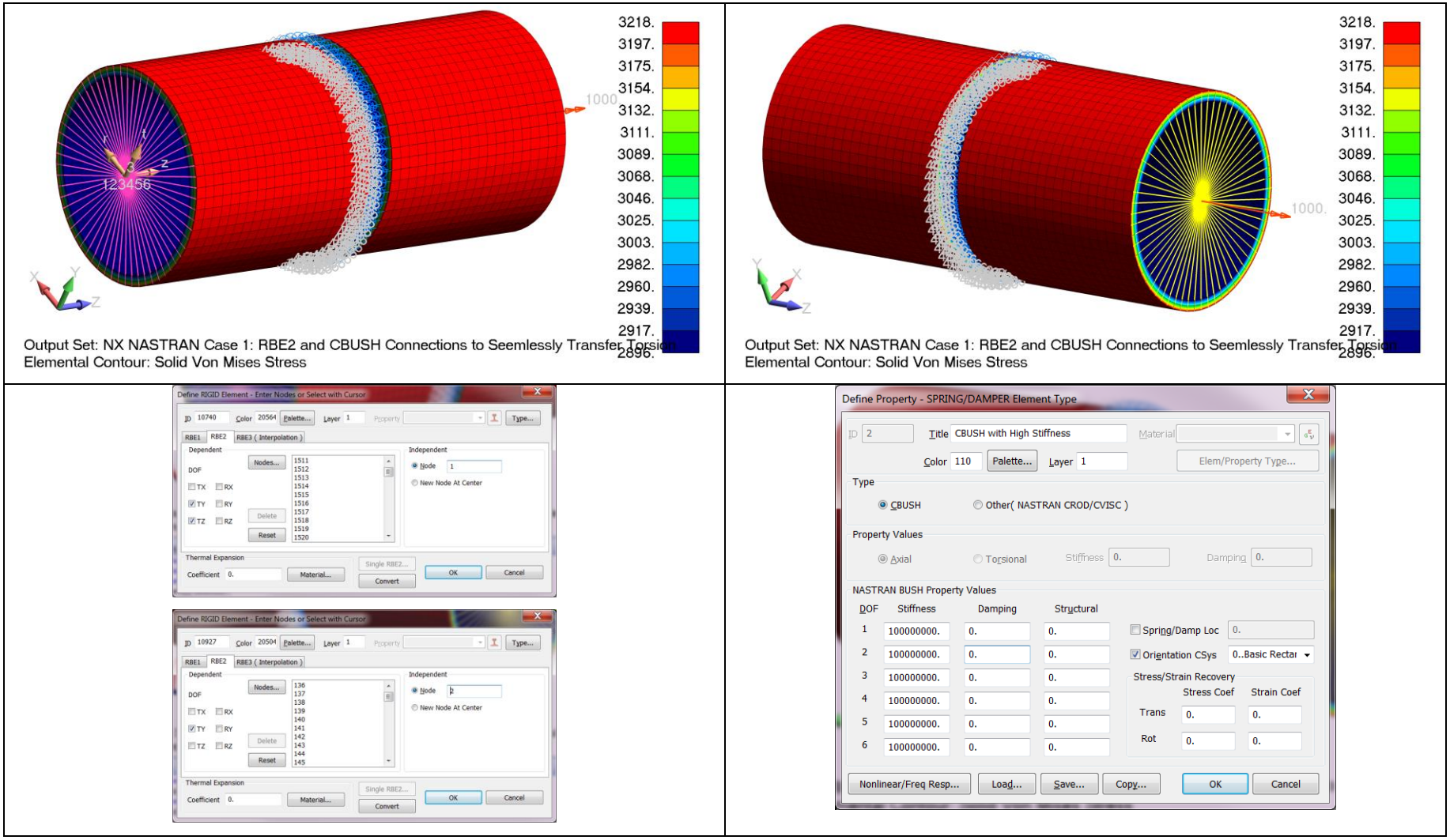


Figure 22: The CBUSH elements with rotational constraints coupled with the RBE2 elements in a cylindrical coordinate system provides a smooth torsion transition along the tube.

All Rights Reserved 2013

4.3 SUMMARY OF CBUSH CHARACTERISTICS

- The CBUSH should be the preferred and automatic element choice when a spring-like element is needed.
- Numerically, it adds stiffness terms directly to the stiffness matrix [K] and as such, user defined stiffness terms should be kept reasonable, i.e., not set to 1.0e99.
- Many advanced features can be enabled for it use with nonlinear and dynamic solutions.
- It should be treated like a beam element with six DOF's. When attached to solid element that only have translational DOF, some care needs to be taken to account for stability and load transfer.
- It is a very useful element to make your models more accurate with less numerical expense.

5. APPENDIX

5.1 THE UM OPTION FOR THE RBE3

Very few analysts avail themselves of this option. As discussed, only independent nodes of a MPC (RBE2 and RBE3) can have SPC's. Since these elements weren't quite complex enough, it was decided to allow the user to apply a constraint to the dependent DOF of the RBE3 element. This ability was added to allow the RBE3 to restrain the structure while simultaneously smearing loads or displacements. In practice, it is most likely rarely used but it exists and therefore merits a bit of an explanation.

Figure 23 shows an example of a cylindrical tube with an applied internal pressure. The tube is simulated as freely expanding. To restrain the free-body-motions of the structure (the six mechanisms of TX, TY, TZ, RX, RY and RZ) without restricting the uniform radial expansion the following sequence of operations was performed:

1. Create a RBE3 with the dependent node at the center of the cylinder and the independent nodes around the circumference;
2. Apply a SPC with all six DOF fixed to the dependent node (should never work but here comes the trick);
3. Create a cylindrical coordinate system with its center at the dependent node of the RBE3 and Modify / Update Other / Output CSys of the independent nodes (for convenience since you really only need to do the nodes within the U_m set);
4. Open up the U_m dialog box within the RBE3 and select the TY and TZ DOF and then three nodes within the independent set equally spaced around the perimeter. This sequence is shown in Figure 23. Note: The reason to use a cylindrical coordinate system allows

All Rights Reserved 2013

the mapping of the TY and TZ onto the Theta and TZ directions of the nodes. When tied together in a set of six DOF, they restrain the six free body mechanisms of the structure. Conceptually this is the hardest step of the process.

5. Analyze and interrogate.

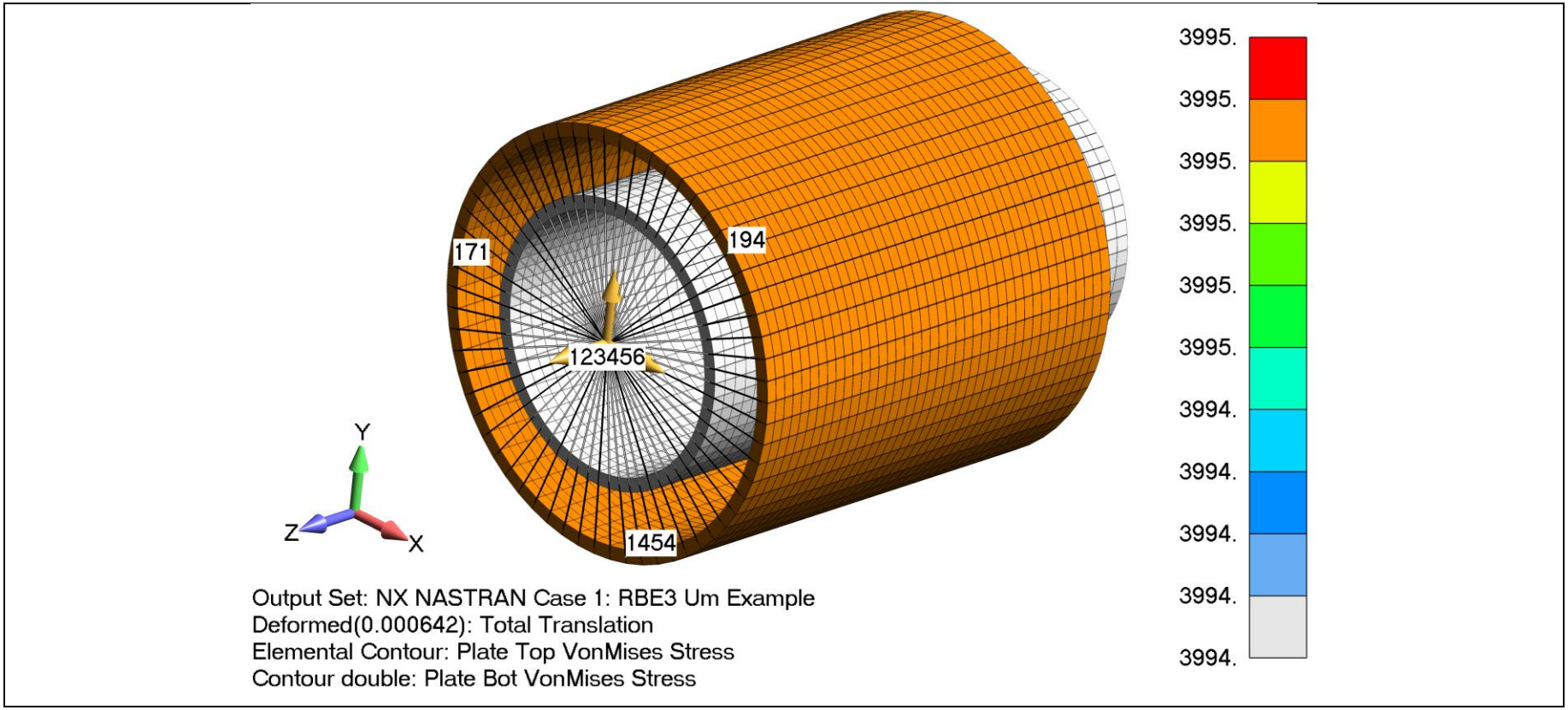


Figure 23: An internal pressure is applied to the cylinder. The RBE3 provides constraint without rigidity. The dependent node is SPC'ed in all six DOF at the center. This is impossible without transferring these SPC's to the U_m DOF.

All Rights Reserved 2013

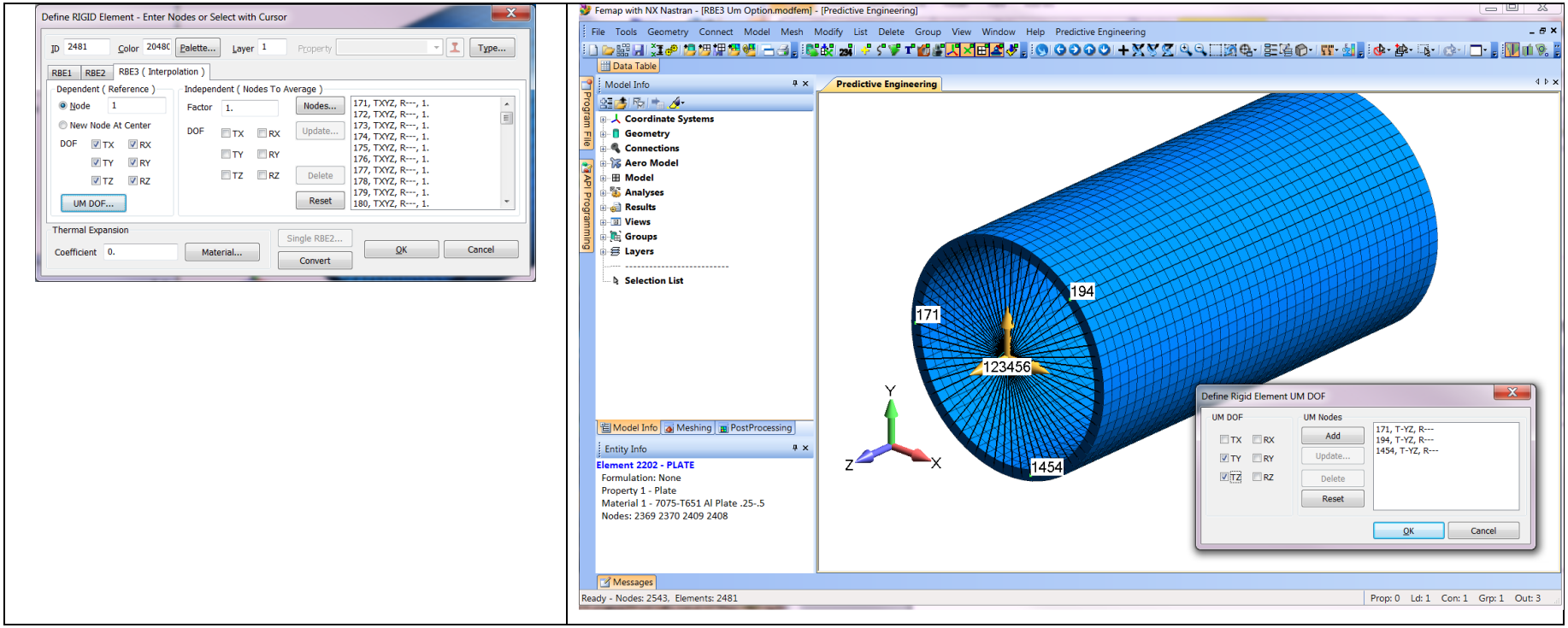


Figure 24: Setting up the RBE3 with the Um option to provide restraint to the internally pressurized cylinder.

5.1.1 WHEN YOU DON'T WANT TO USE THE RBE3 UM OPTION

We would be remiss not to show that this same problem can be solved using a RBE2 element. A cylindrical coordinate system is still employed but the dependent DOF's that are engaged are only the TY ad TZ. Results from this model are shown in Figure 25. This is a unique example and of course there are situations where the Um option would be useful but in this particular example there are options.

All Rights Reserved 2013

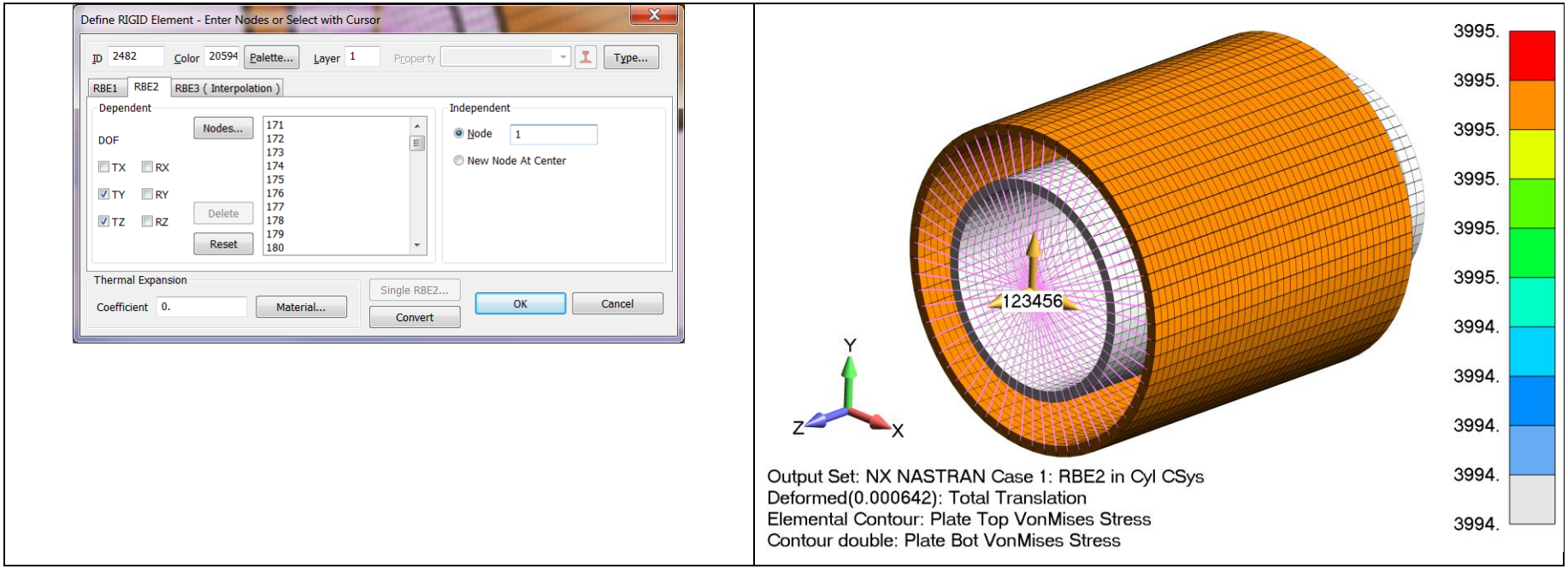


Figure 25: Another approach to solving the internal pressurized cylinder with an RBE2 connection.

5.1.2 WHAT ABOUT THE CBUSH?

Since we are beating this example to death, yes, one could get the same result using the CBUSH. The important trick is to set the CBUSH Orientation CSys into the user defined cylindrical coordinate system defined for the RBE3 and RBE2 examples. Figure 26 shows the results from this application.

All Rights Reserved 2013

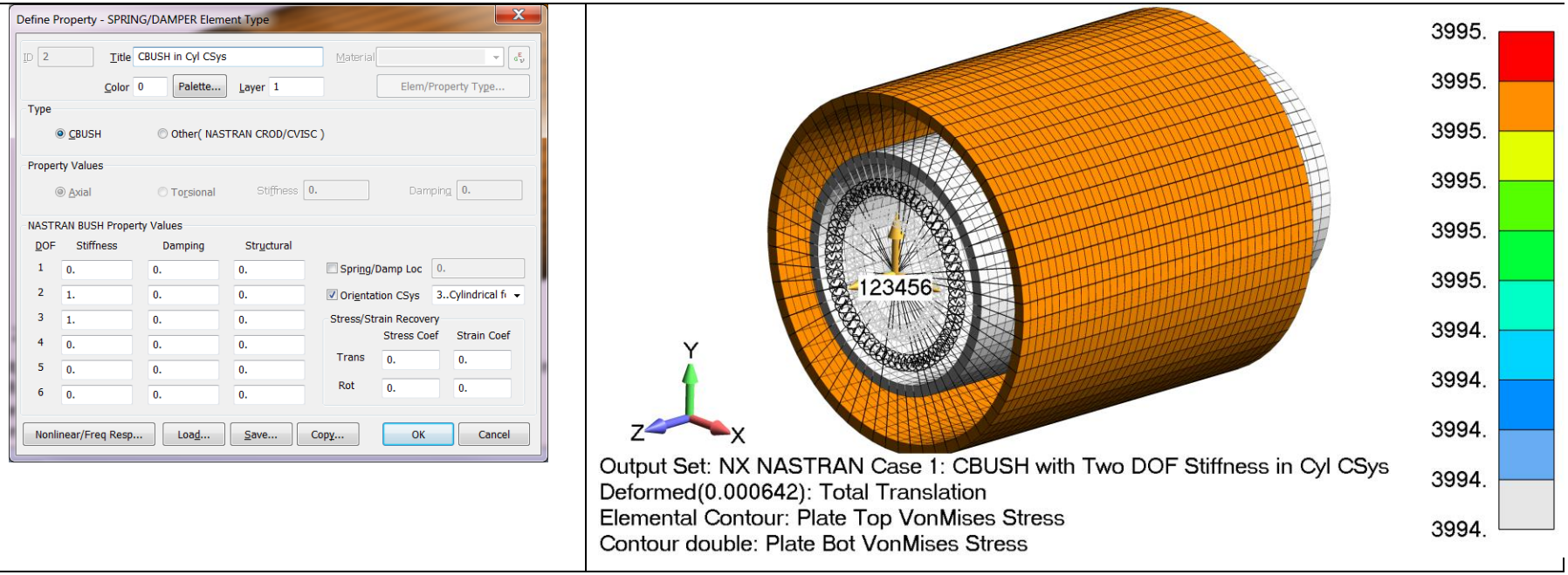


Figure 26: Using the CBUSH Element to obtain the same result as shown for the RBE3 and RBE2 connections.

All Rights Reserved 2013