
# Technical Seminar:
# Connection Elements - The Boring Stuff

Topics:

- Building Expert-Class FE Models
- The Basics: RBE Elements (MPC Elements)
- The standard in Spring Elements: CBUSH
- Femap Tips and Tricks of the Moment: Resolution, Turning off References, Setting View Display Options

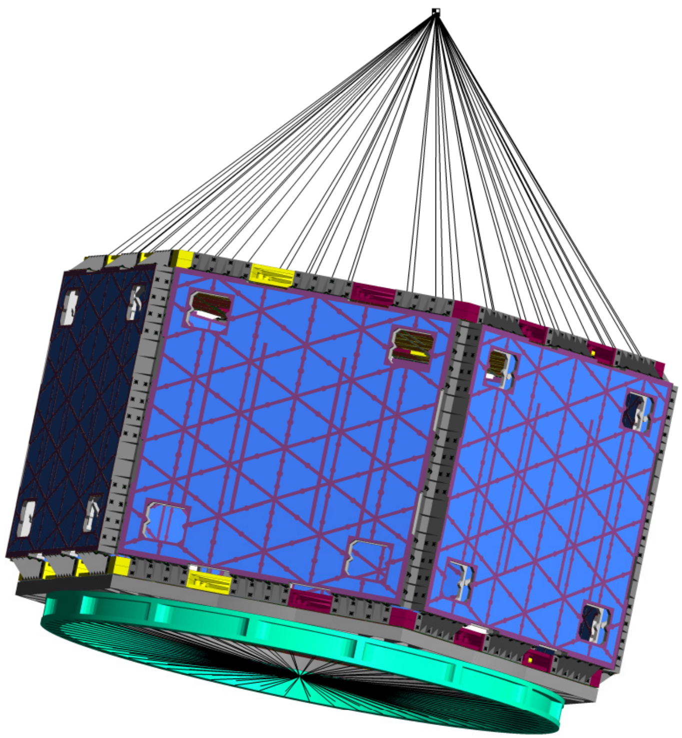
A Brief Q&A Period

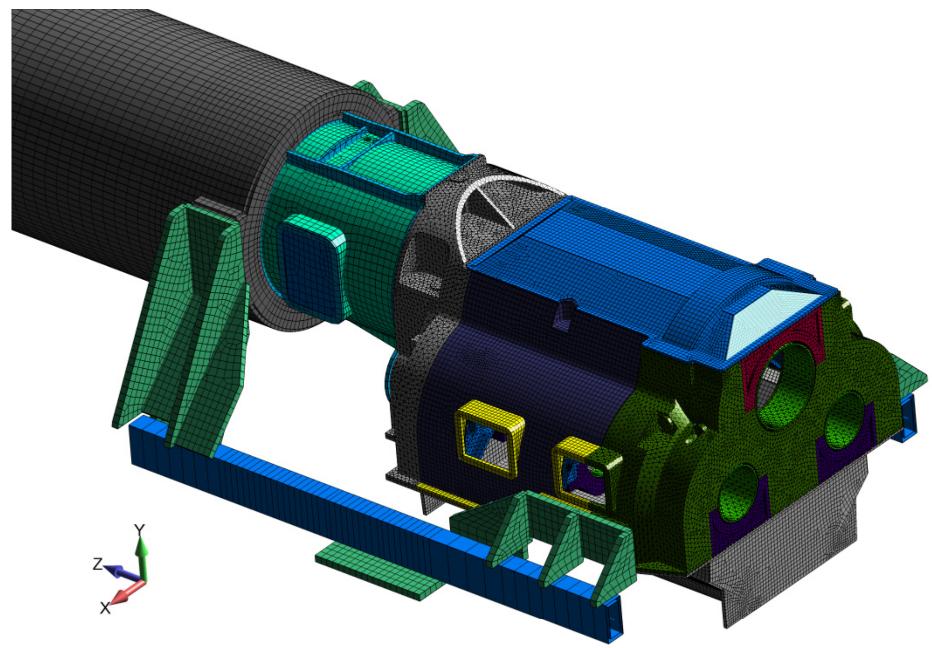
April 2011

FINITE ELEMENT ANALYSIS
Predictive Engineering


# Technical Seminar: Efficient Connections

Building an expert-class FE model is often all about creating efficient connections.






# Technical Seminar: Efficient Connections

## The Basics: R-Type Elements {Rigid Body (RBE2) and *Not* Rigid Body (RBE3)}

## A very useful resource: *NX Nastran Element Library Reference*

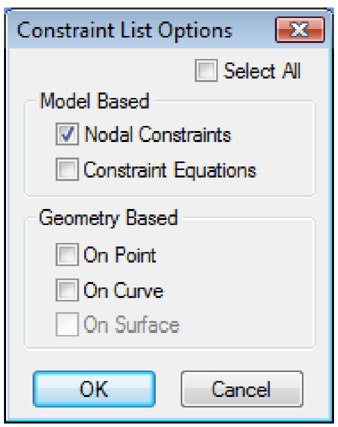
**Table 7-1. Rigid Element and MPC Entries**

| Name | Description | $m$ = Dependent Degrees-of-freedom |
|---|---|---|
| RROD | A open-ended rod which is rigid in extension | $m = 1$ |
| RBAR | Rigid bar with six degrees-of-freedom at each end. | $1 \le m \le 6$ |
| RTRPLT | Rigid triangular plate with six degrees-of-freedom at each vertex. | $1 \le m \le 12$ |
| RBE1 | A rigid body connected to an arbitrary number of grid points. The independent and dependent degrees-of-freedom can be arbitrarily selected by the user. | $m \ge 1$ |
| RBE2 | A rigid body connected to an arbitrary number of grid points. The independent degrees-of-freedom are the six components of motion at a single grid point. The dependent degrees-of-freedom at the other grid points all have the same user-selected component numbers. | $m \ge 1$ |
| RBE3 | Defines a constraint relation in which the motion at a "reference" grid point is the least square weighted average of the motions at other grid points. The element is useful for "beaming" loads and masses from a "reference" grid point to a set of grid points. | $1 \le m \le 6$ |
| RSPLINE | Defines a constraint relation whose coefficients are derived from the deflections and slopes of a flexible tubular beam connected to the referenced grid points. This element is useful in changing mesh size in finite element models. | $m \ge 1$ |
| RSSCON | Define a multipoint constraint relation which models a clamped connection between shell and solids. | $m \ge 5$ |
| MPC | Rigid constraint that involves user-selected degrees-of-freedom at both grid points and at scalar points. The coefficients in the equation of constraint are computed and input by the user. | $m = 1$ |


# Technical Seminar: Efficient Connections

## The Basics: SPC and MPC Constraints (What is under the hood of RBE's)

*NX Nastran User Guide: Chapter 9 Constraints*




### 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.

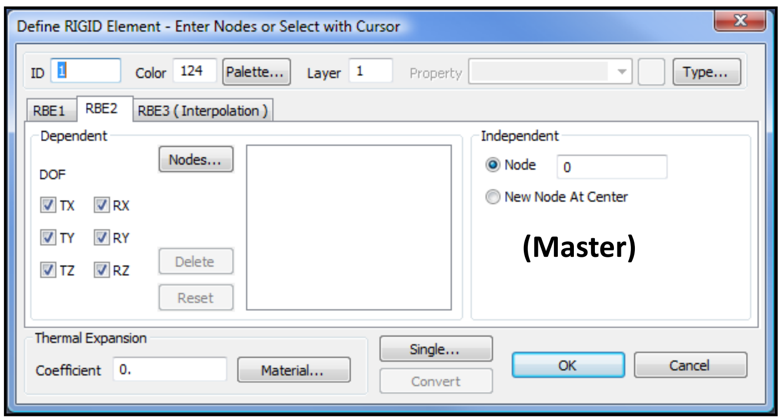
Real world structures often don't have simple or ideal boundary conditions. Because a model's constraints greatly influences its response to loading, you must try to constrain your model as accurately as possible.

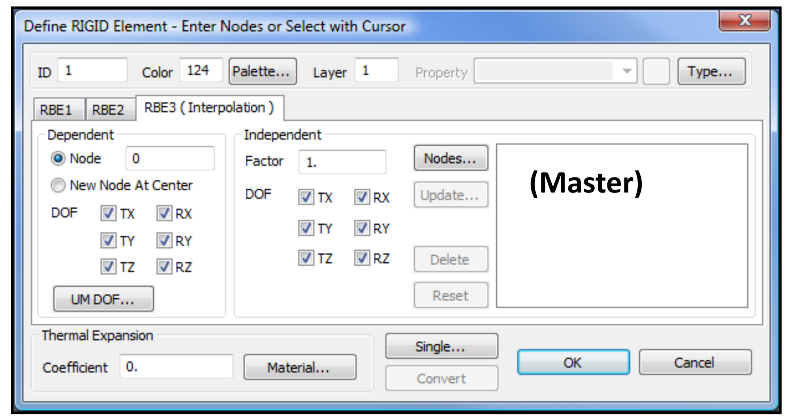
This chapter describes how you apply constraints. To understand how constraints are processed, you need to be familiar with the NX Nastran set notation and matrix operations.


## Technical Seminar: Efficient Connections

# The Basics: R-Type Elements {RBE2 & RBE3}

The RBE2 element doesn't cause numerical difficulties because it doesn't add terms to the stiffness matrix. The RBE2 element is actually a constraint element that prescribes the displacement relationship between two or more grid points. This is the same concept used by the RBE3. The central theme is Dependent versus Independent nodes.




April 2011

**Note:**

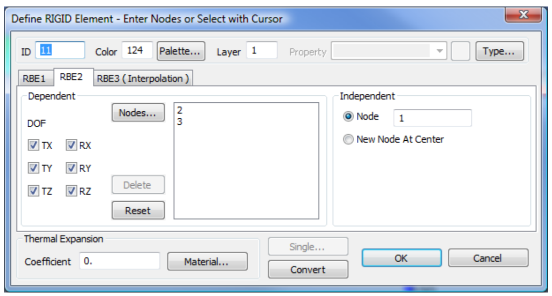
Using rigid elements will cause incorrect results in buckling and differential stiffness analyses because the large displacement effects are not calculated. ***Exceptions*** are zero length elements (e.g.,to simulate a hinge) and rigid elements constrained so that they don't rotate.

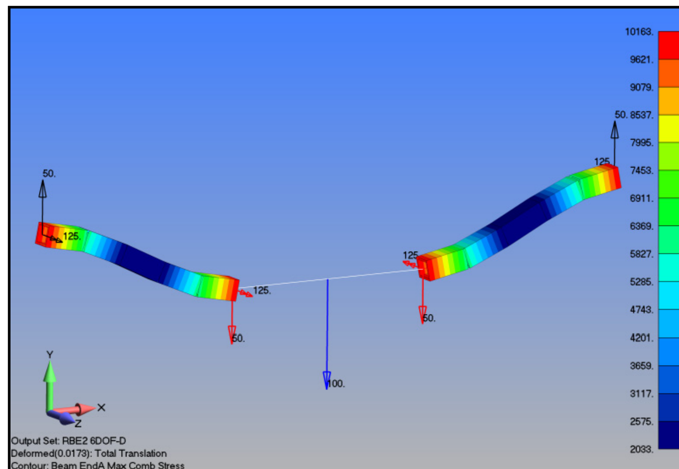
Finite Element Analysis
Predictive Engineering


# Technical Seminar: Efficient Connections

## The Basics: R-Type Elements {RBE2}

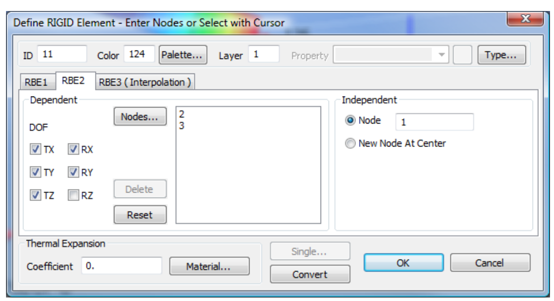
Here is a starting point to understand the RBE2. A very simple beam model with a 100 lbf load at the center of the beam. The beam is held together with a RBE2.

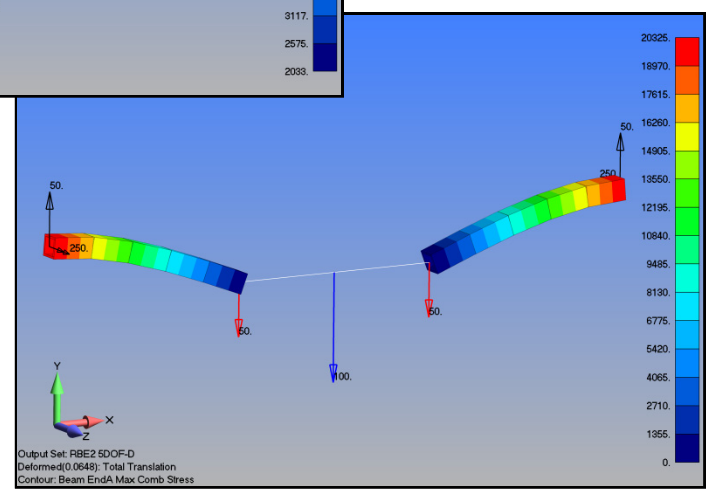



*Moment & Force Calc.*
*M = 2.5*50*
*F = 100/2*

- **Simple Beam with fixed ends**
- **Turn On Equation Force**
- **Free-Body-Diagram**
- **Inspect F06 (MPCForce)**






Finite Element Analysis
Predictive Engineering

Model File: First RBE2 Model.modfem

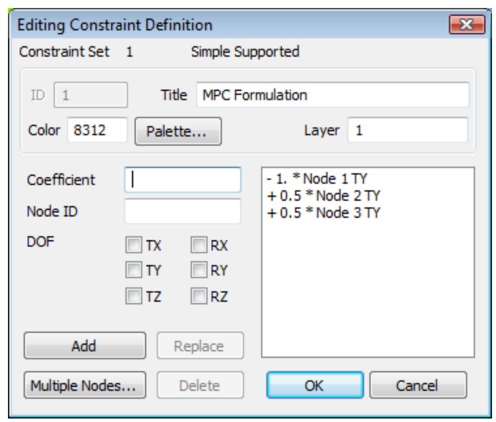

# Technical Seminar: Efficient Connections

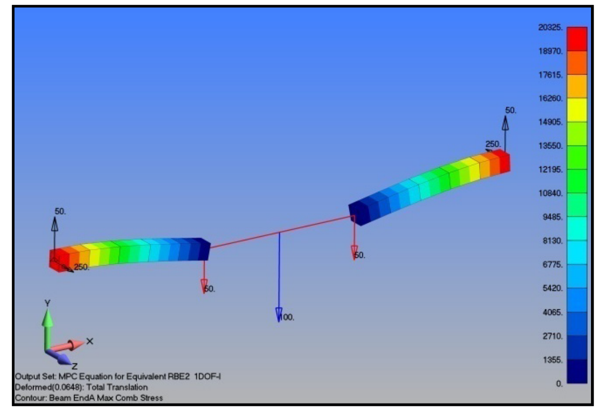
## The Basics: R-Type Elements {RBE2} and MPC's {Constraint Equations}

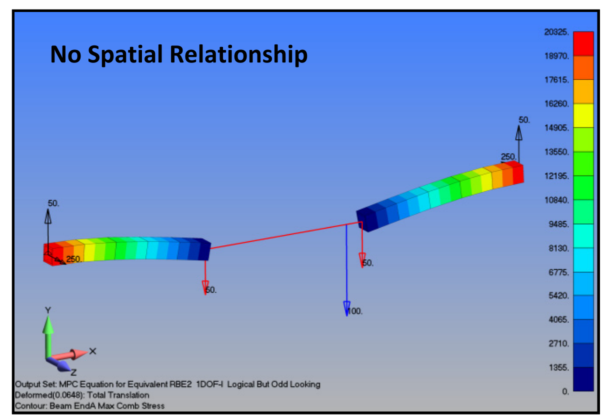
The underlying numerics to RBE's is the constraint equation. One can be thankful that we don't have to enter them by formula but it is good to see how they work.

- All RBE's are just MPC's
- The first term is the dependent node ($a_D$) and depends upon the movement of the other independent (master) nodes.
- Setting up MPC's requires one to understand the basic equation of mechanics.

$$\underline{a_D X} = a_1 X_1 + a_2 X_2 + a_3 X_3 + \ldots + a_n X_n$$




Model File: Constraint Equations.modfem


# Technical Seminar: Efficient Connections / Numerical Foundation

## The Basics: MPC's (i.e., RBE's) are based on constraint equations.

- The dependent nodes are removed from the displacement formulation matrix since their behavior is described by the independent nodes.
- Why do we care? (i) One can't apply displacements to dependent nodes since they no longer exist within the "u" column; (ii) However, Forces/Moments can still be applied to dependents and independents; (iii) matrix formulations are used and inverted. Hence your RBE must ***not*** have a mechanism. RBE2's have 6 DOF for the independent node while RBE3's are defined by the user (hence mechanisms can occur!).
- By using MPC's, NX Nastran creates numerically perfect Rigid's / Interpolation / Etc.

**NX Nastran**

**Nastran Theoretical Manual**

3.5.1 Multipoint Constraints

The multipoint constraint equations are initially expressed in the form,

$$[R_g]\{u_g\} = 0, \qquad (1)$$

where the coefficients are supplied by the user. The user also specifies the degree of freedom that is made dependent by each equation of constraint, so that the $\{u_g\}$ matrix may immediately be partitioned into two subsets,

$$\{u_g\} = \left\{\begin{array}{c} u_n \\ \hline u_m \end{array}\right\} , \qquad (2)$$

where the set, $u_m$, is the set of dependent degrees of freedom. The matrix of constraint coefficients is similarly partitioned

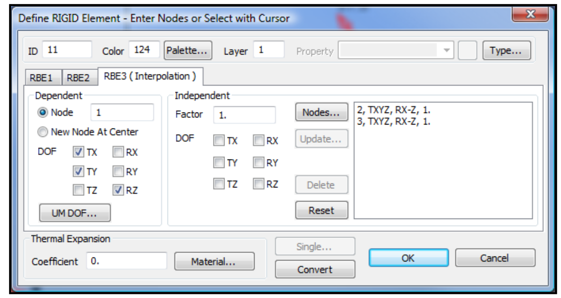
$$[R_g] = [R_n \mid R_m] , \qquad (3)$$

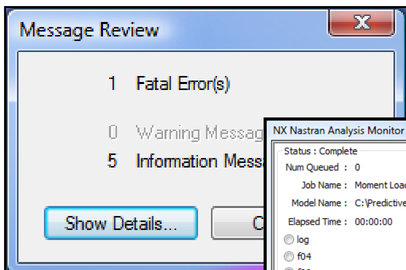
3.5-1

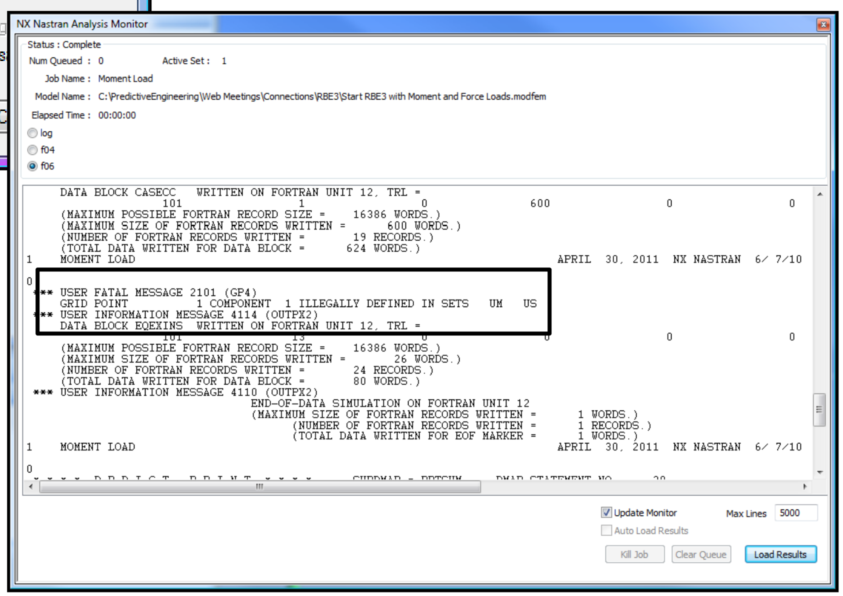

# Technical Seminar: Efficient Connections / Numerical Foundation

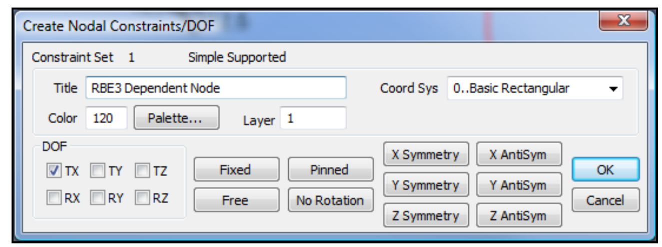
## The Basics: MPC's (i.e., RBE's) are based on constraint equations.

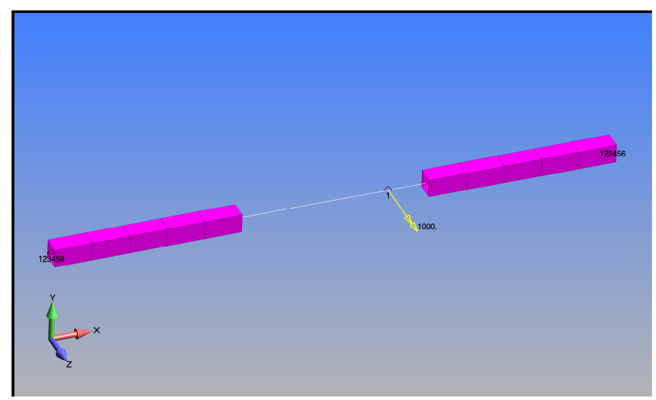
- What happens when you constrain a dependent DOF of a RBE2 or RBE3? From the mechanics we know that the solver has to be unhappy. This is how it tells us:



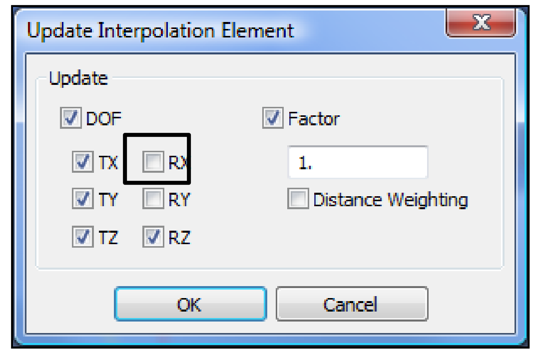
April 2011

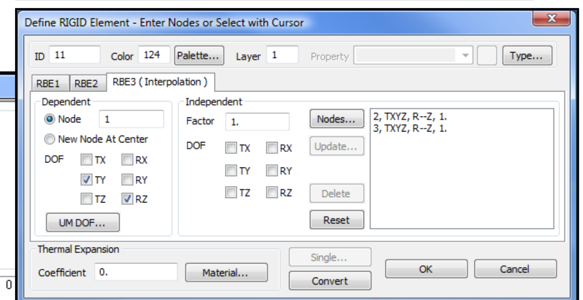
FINITE ELEMENT ANALYSIS
Predictive Engineering

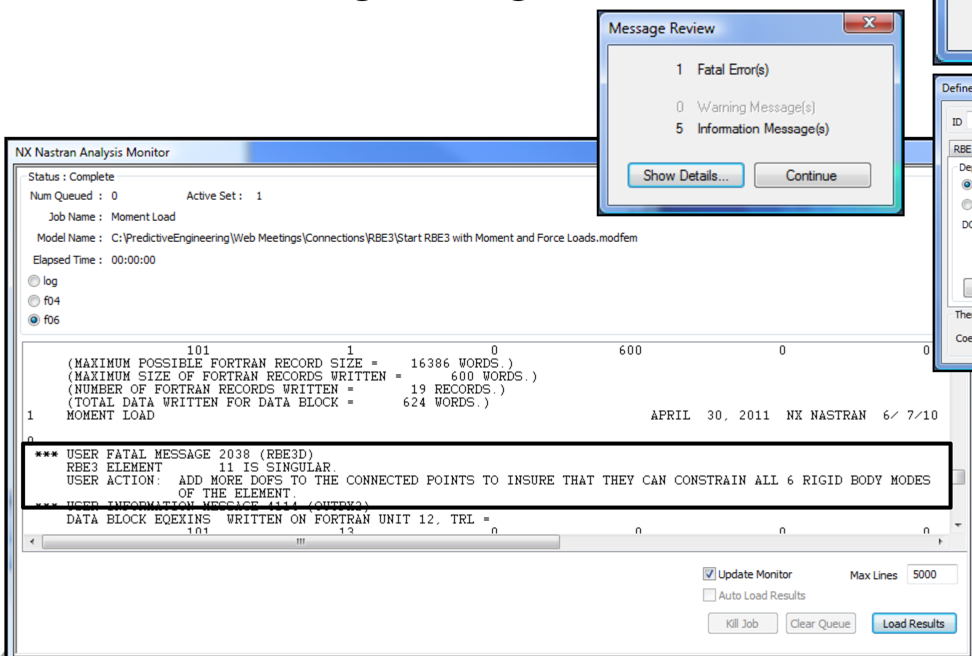

# Technical Seminar: Efficient Connections / Numerical Foundation

## The Basics: MPC's (i.e., RBE's) are based on constraint equations.

- What happens when your RBE3 has a mechanism? Here we remove the stability DOF (RX) from our RBE3 in the prior model.
- Please be careful and don't over constrain since there are no "warning messages".



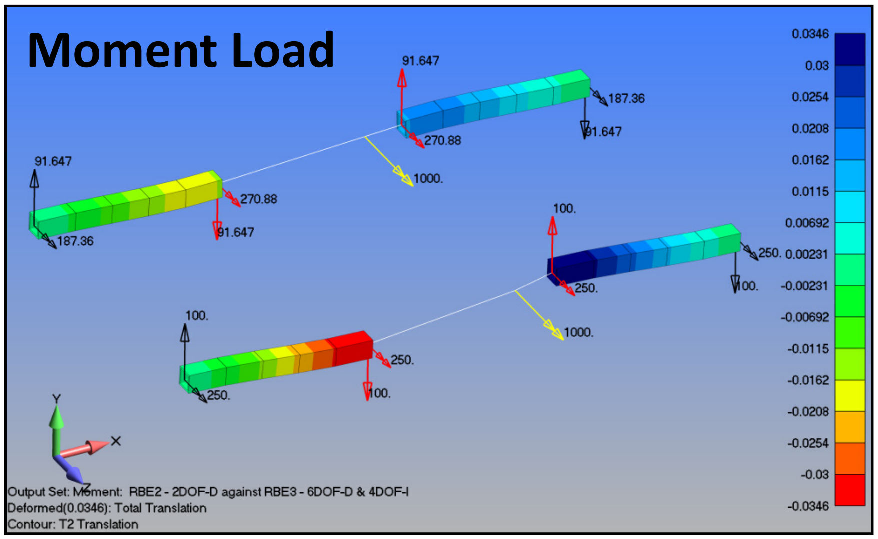



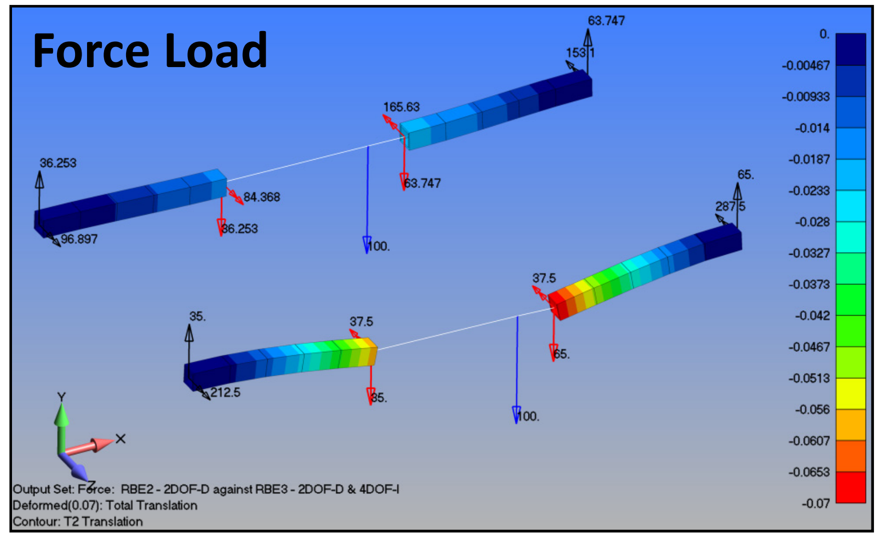

**Note** that the error message calls out Element 11. NX Nastran tries to help fix the problem as much as it can.


# Technical Seminar: Efficient Connections

## The Basics: R-Type Elements {RBE2 &RBE3}

A comparison is presented between RBE2 and RBE3 formulations. Most people can figure out the load distribution for the RBE2 element but that for the RBE3 defies simple logic if you don't know the mathematical formulation.




Model File: Start RBE3 with Moment and Force Loads.modfem

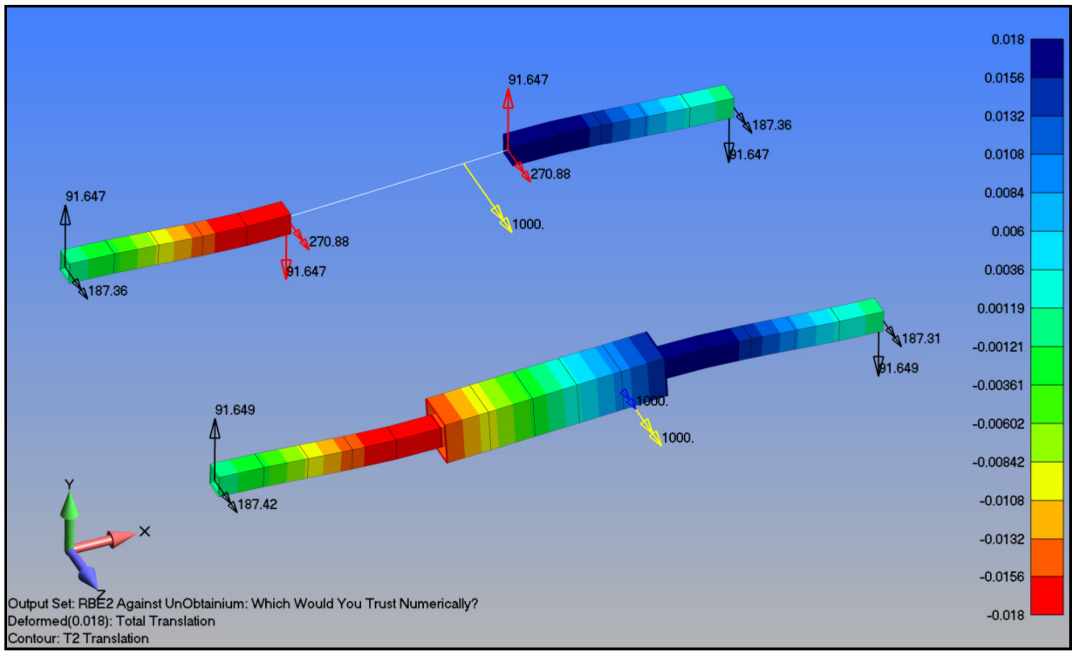
***RBE2 Moment Calculation***
***M = 2*270.88 + 2*2.5*91.647***
***M = 541.76 + 458.24 = 1,000***
***But why the different numbers? See next slide…….***

**Note:** For the RBE3 element we must include the TX, TZ, & RX DOF on the *Independent* side to prevent rigid body rotation of the RBE3 element. This is *tricky*…see prior slide on Numerical Foundation.


# Technical Seminar: Efficient Connections

## The Basics: R-Type Elements {RBE2 and Super-Stiff}

Understanding how the RBE2 works can be illustrated by comparing it against a super-stiff beam.




***RBE2 Moment Calculation***
***You really can't do with simple Force and Moment Summations.***
***The RBE2 does a perfect job of enforcing the exact rigid constraint.***



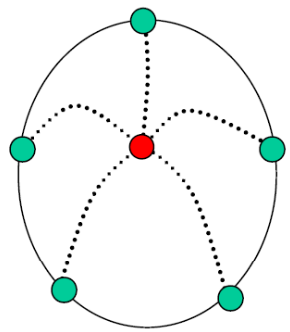
Model File: RBE2 and Super-Stiff Beam with Moment and Force Loads.modfem


## Technical Seminar: Efficient Connections

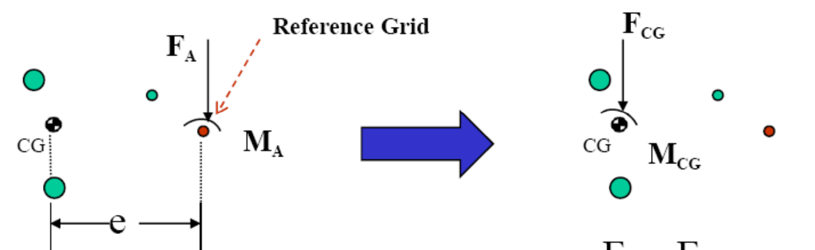
### The Basics: R-Type Elements {RBE3} – Mechanics

The RBE3 element does not create stiffness in the structure. The trick is understanding how the applied force and moment is distributed over the master (independent) nodes.

- **Motion at a dependent GRID is the weighted average of the motion(s) at a set of master (independent) GRIDs**



  - **NOT a "rigid" element**
  - **IS an interpolation element**
  - **Does not add stiffness to the structure (when used correctly)**



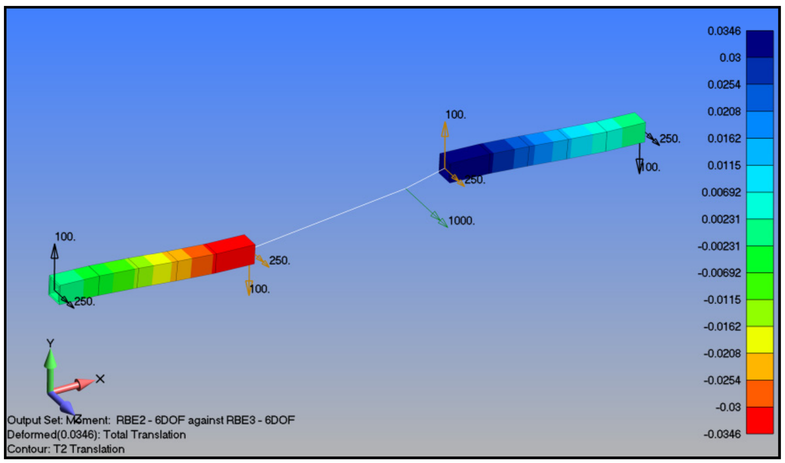

$$F_{CG} = F_A$$

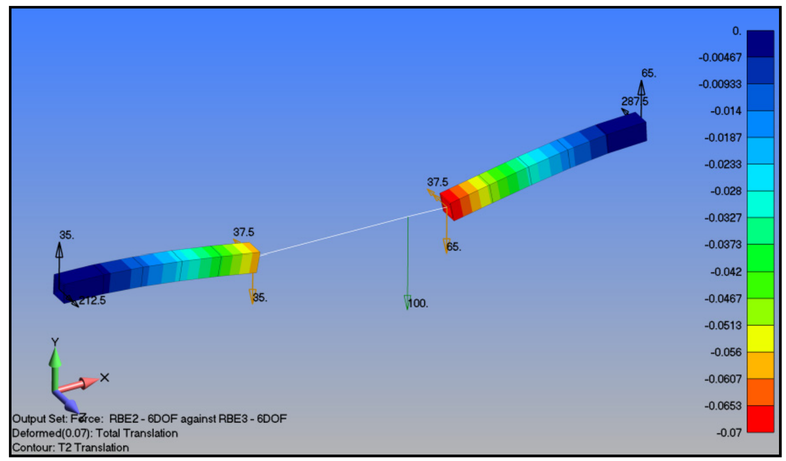
$$M_{CG} = M_A + F_A * e$$

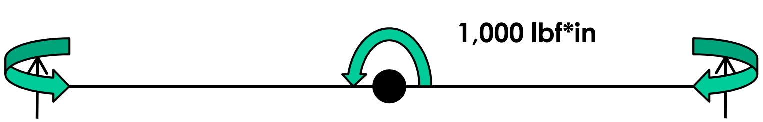

# Technical Seminar: Efficient Connections

## The Basics: R-Type Elements {RBE3}

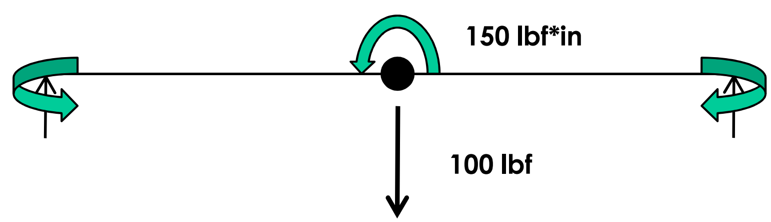
Here is the background mechanics on the RBE3 element. You take the force/moment and apply it at the centroid of the independent (master) nodes and then split it up equally.




$\Sigma M = 2*250 + 2*100*2.5 = 1{,}000$ lbf*in



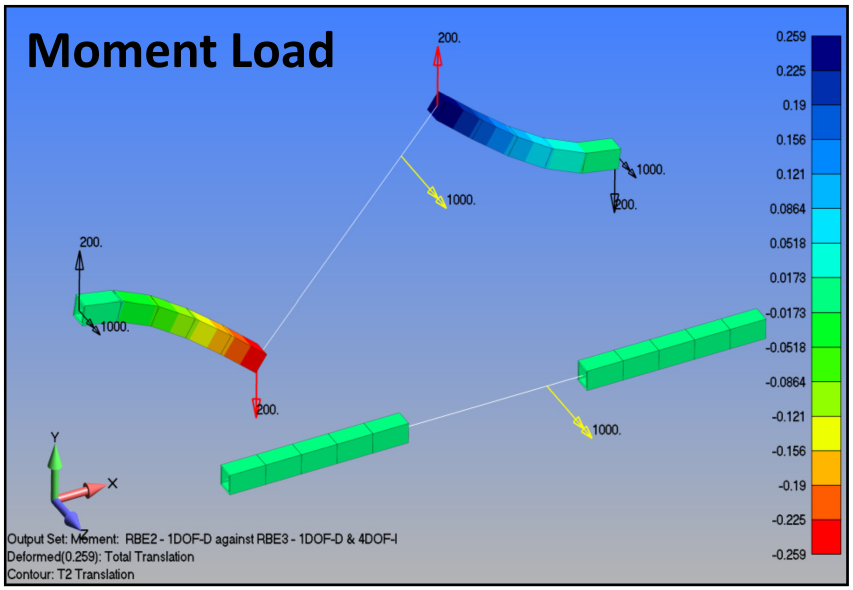

$\Sigma M = 2*37.5 + 15*2.5 + 15*2.5 = 150$ lbf*in

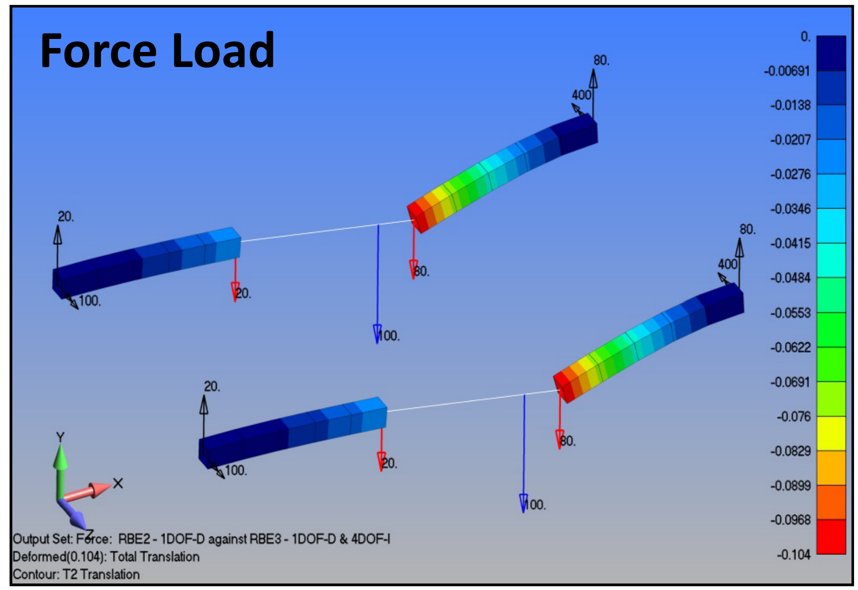
$\Sigma F = (50 - 15) + (50 + 15) = 100$ lbf


# Technical Seminar: Efficient Connections

## The Basics: R-Type Elements {RBE3}

If we reduce the Dependent DOF of to TY (1DOF), then the RBE2 and RBE3 calculate the following:




**When the RBE3 has no RZ – you have no RZ**



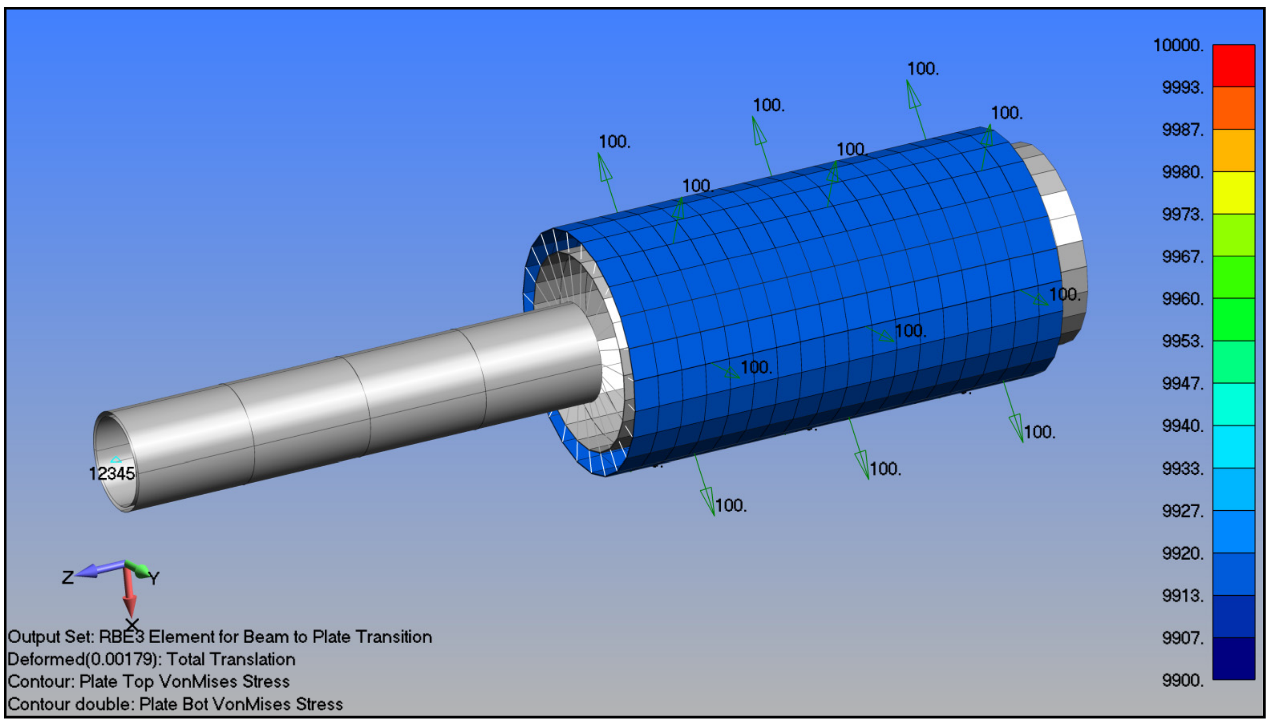


Final Example: Final Example RBE3 with Moment and Force Loads.modfem


# Technical Seminar: Efficient Connections

## An Example: R-Type Elements {RBE3}

A pipe structure is modeled using plate elements. It is part of a larger structure that is then connected to a beam element. By using a RBE3 element we can make a graceful transition before the beam and plate elements. It sounds simple but it can also be tricky to implement.




*The pipe expands and contracts due to poisons effect.*

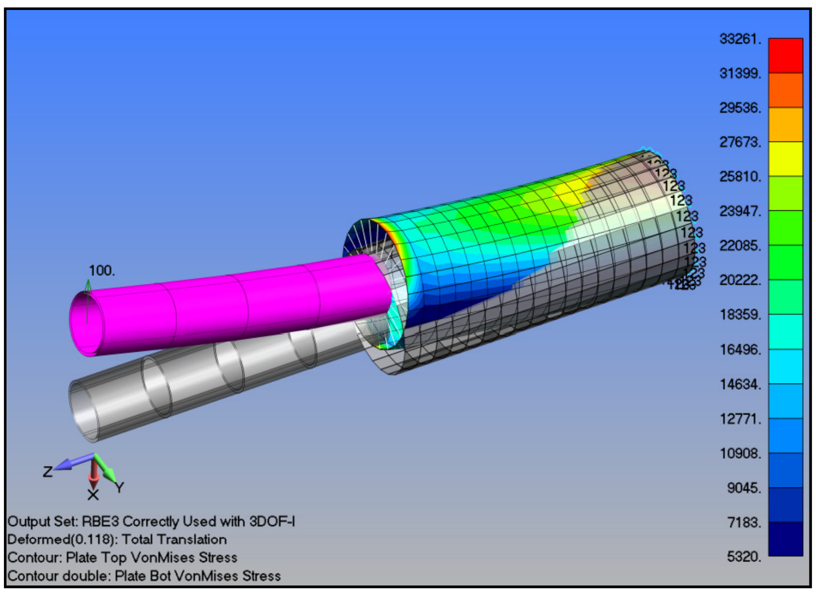
Model File: RBE3 Plate and Tube Model.modfem

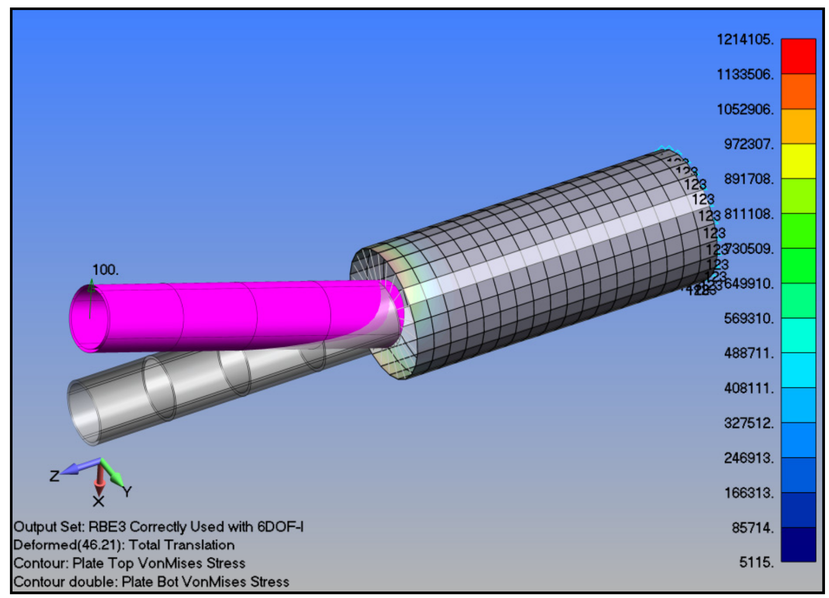

## Technical Seminar: Efficient Connections

### RBE3 Usage Guidelines:

- **Do not** specify rotational DOF for Independent Nodes **except** when necessary to avoid singularity caused by a linear set of master grids (Remember Prior Beam Model)

- Using rotational DOF on Independent grids can result in implausible results



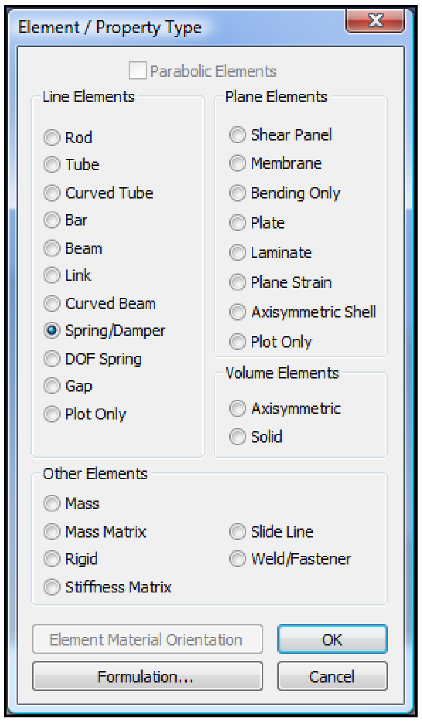


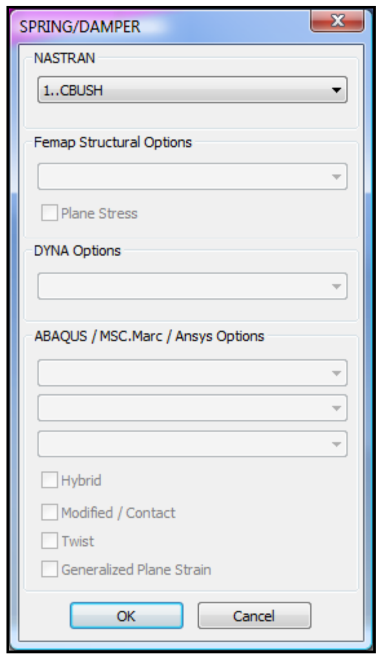

Model File: RBE3 Plate and Tube Model Guidelines.modfem

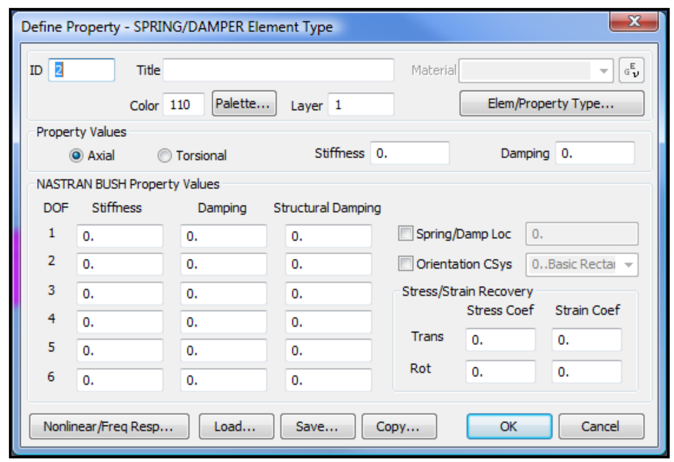



# CBUSH Elements: Perfect Spring Connection

Setting up the Element Property for the CBUSH requires special care since it is not the default formulation within the Femap menu structure.




# Technical Seminar: Efficient Connections

## CBUSH Elements: NX Nastran Element Library – Chapter 6

**Chapter 6** *Special Element Types*

### 6.2 Bushing (CBUSH) Elements

The bushing (generalized spring and damper) elements consist of the following:

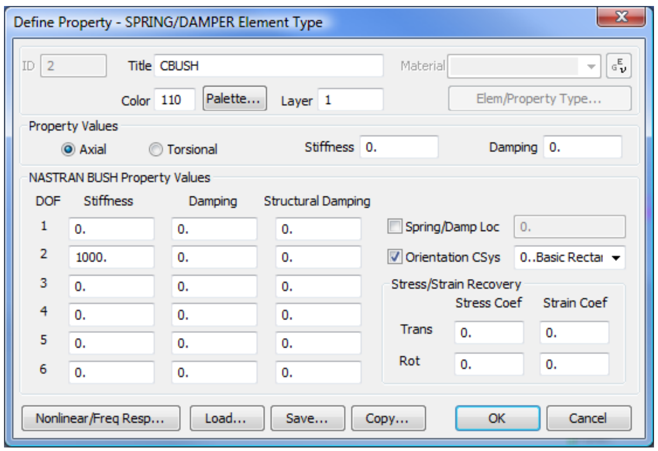
- CBUSH
- CBUSH1D

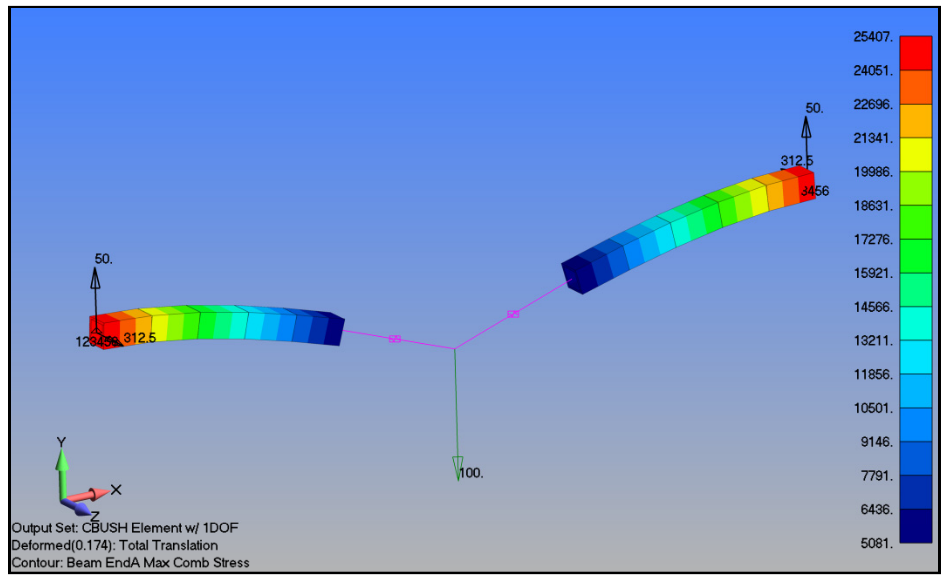
The generalized spring-damper element CBUSH is a structural scalar element connecting two noncoincident grid points, or two coincident grid points, or one grid point with an associated PBUSH entry. This combination is valid for any structural solution sequence. To make frequency dependent the PBUSH need only have an associated PBUSHT Bulk Data entry. The PBUSHT entry for frequency dependency is only used in SOL 108 and SOL 111. You can also use the PBUSHT entry to define load-displacement dependency in SOL 106.

Figure 6-3 shows some of the advantages of using the CBUSH element over CELASi elements. For example, if you use CELASi elements and the geometry isn't aligned properly, internal constraints may be induced. The CBUSH element contains all the features of the CELASi elements plus it avoids the internal constraint problem. The following example demonstrates the use of the CBUSH element as a replacement for scalar element for static analysis. The analysis joins any two grid points by user-specified spring rates, in a convenient manner without regard to the location or the displacement coordinate systems of the connected grid points. This method eliminates the need to avoid internal constraints when modeling.


# Technical Seminar: Efficient Connections

CBUSH Elements: Is a 6DOF customizable pseudo-beam element. It has all the advantages of a beam element with the ability to dial in spring values.

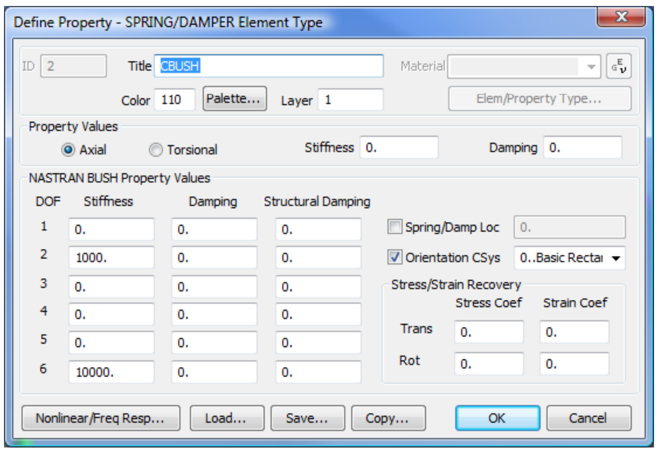


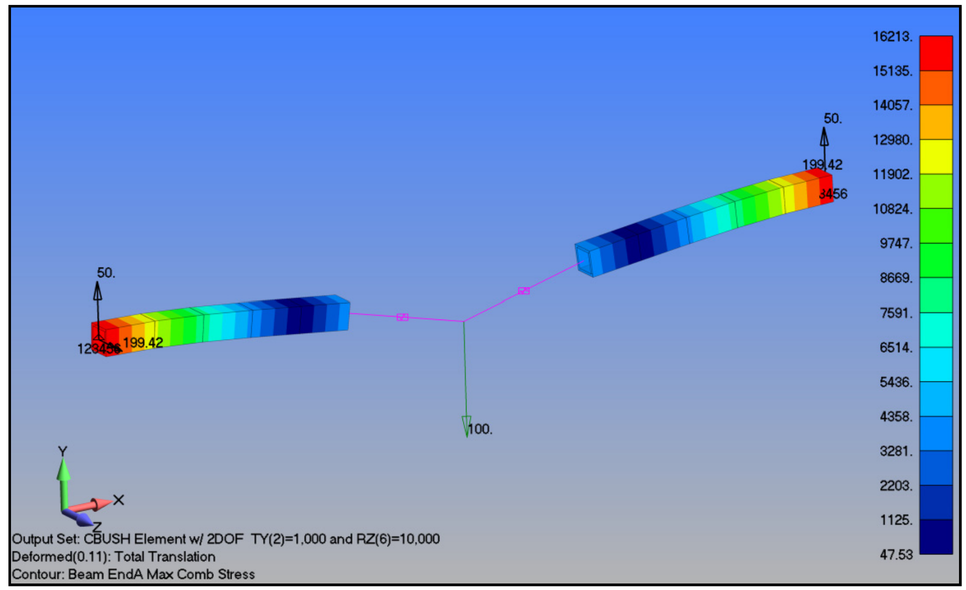




**Note:** If the CBUSH element is not properly constrained, it will create mechanisms much like any other element. This example works well since the beam element that the CBUSH is attached to has 6DOF and is fully constrained.


# Technical Seminar: Efficient Connections

CBUSH Elements: Is a 6DOF customizable pseudo-beam element. It has all the advantages of a beam element with the ability to dial in spring values.



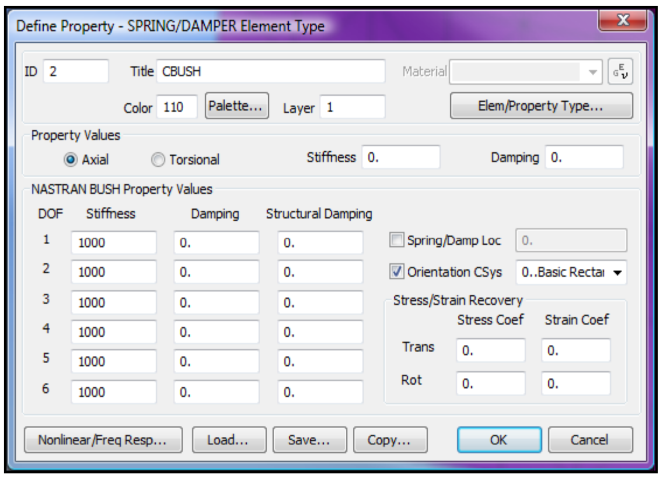



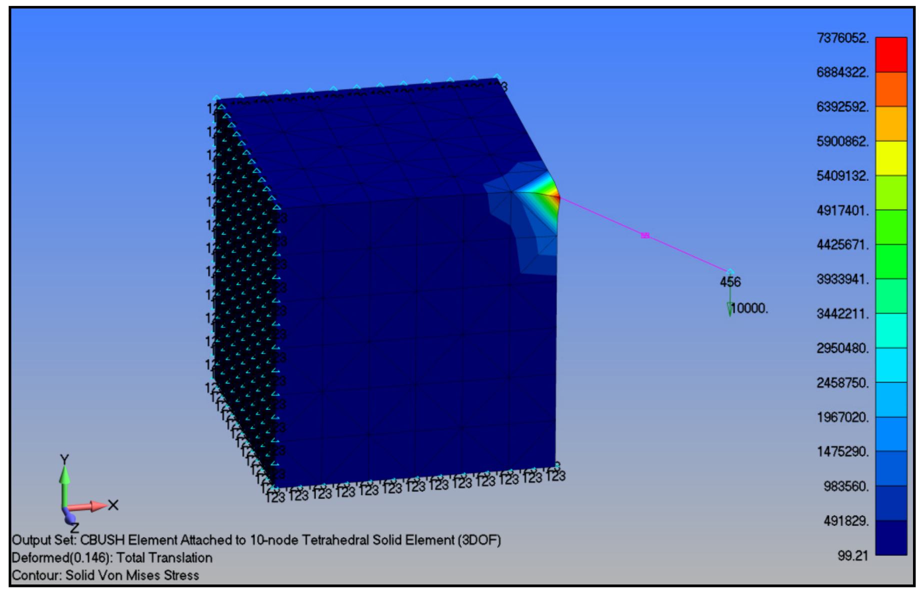



Finite Element Analysis
Predictive Engineering


# Technical Seminar: Efficient Connections

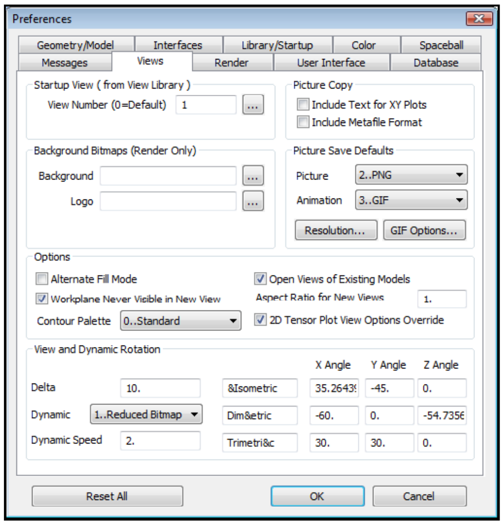
CBUSH Elements: Connecting CBUSH elements (and likewise beam elements) to 3DOF solid elements can be problematic due to rigid body motions (mechanisms).

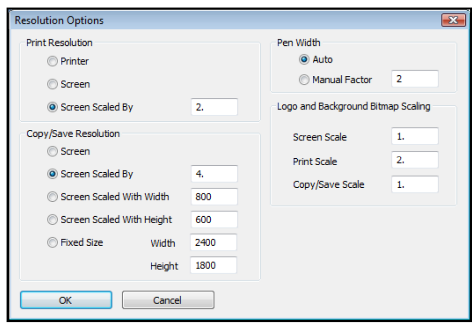


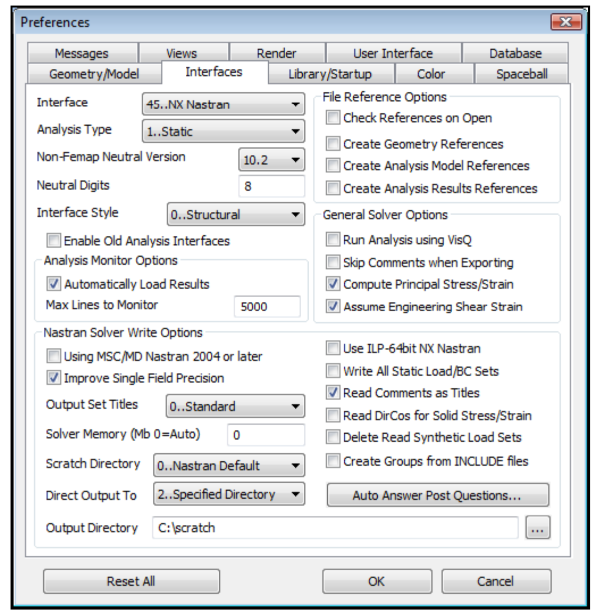




# Technical Seminar: Efficient Connections

## My Standard Femap Settings:
## High-Resolution Graphics and Housekeeping



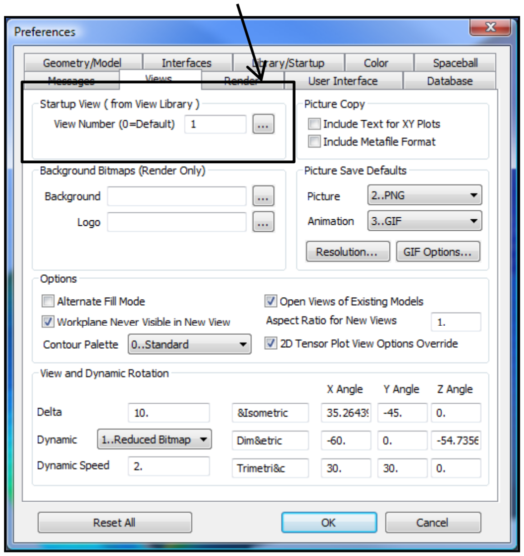



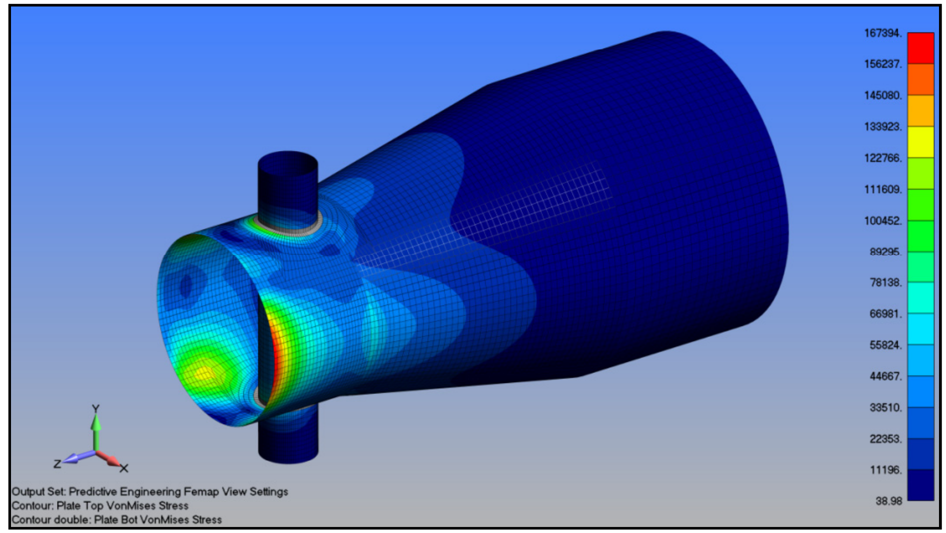

April 2011

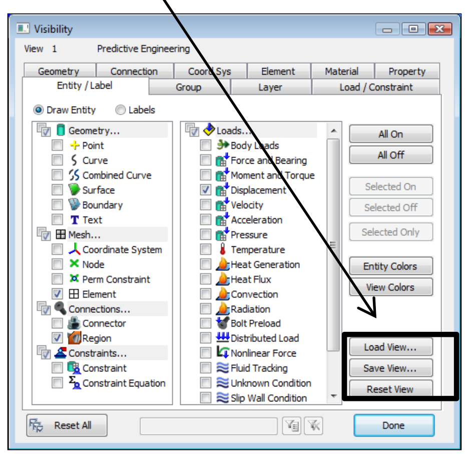

# Technical Seminar: Efficient Connections

## My Standard Femap Settings: View Settings

- View Settings Can Be Saved
- Think database
- Think "do once"




Open Model File: Predictive Engineering Recommended View Settings.modfem