

# An Introduction to Tolerance Analysis of Flexible Assemblies

K. G. Merkle, K. W. Chase, E. Perry  
Brigham Young University  
Provo, UT

## **Abstract**

Tolerance analysis is used to predict the effects of manufacturing variation on finished products. Either design tolerances or manufacturing process data may be used to define the variation. Current efforts in tolerance analysis assume rigid body motions. This paper will present a method of combining the flexibility of individual parts, derived from the finite element method, with a rigid body tolerance analysis of the assembly. These results can be used to predict statistical variation in residual stress and part displacement. This paper will show that manufacturing variation can produce significant residual stress in assemblies. It will demonstrate two different methods of combining tolerance analysis with the flexibility of the assembly.

## **Introduction:**

Tolerance analysis is the process of determining the effect that the tolerances on individual manufactured parts will have on an assembly of these parts. Tolerance analysis is a subset of Design for Assembly (DFA) and Design for Manufacturability (DFM). As such, tolerance assignment forms an important link between the design and manufacturing processes. Tolerance variation in an assembly is derived from three major sources: size variation, geometric variation, and kinematic variation. Size variation occurs due to the variability of the dimensions. Geometric variation occurs due to variations in form, such as flatness or cylindricity. Kinematic variation occurs as small adjustments between mating parts in response to dimensional and geometric variations. As parts are assembled the tolerances in each part add together to form “tolerance stack-up”. The result is that many small tolerance variations can add together to form a large residual stack-up, which can affect product performance and cost.

Unfortunately, designers often view tolerance assignment as either a “black art” that they don't understand or as a trivial part of the total design. With the increasing emphasis on DFA/DFM, these views become untenable. To overcome this kind of thinking, engineers must be provided with tools that will allow them to understand the consequences of tolerance assignment and their relationship to product performance. This paper will provide an overview of a tolerance analysis package that is integrated into the design process and show how the tolerance information can be coupled with MSC/PATRAN [1] and MSC/NASTRAN [2] to predict assembly stresses in flexible parts. It will develop a linear method of solving certain contact problems, define the limitations of this method, and show two different methods of implementing this method in MSC/NASTRAN for calculating assembly stresses due to tolerance stack-up. This methodology can provide engineers and designers with a useful measure of the effect of manufacturing tolerances early in the design process.

## **Previous Research:**

Tolerance analysis has been a fertile ground for research. Some of the early researchers in solid modeling such as Hillyard, Braid [3] and Requicha [4][5] have all looked at the tolerancing problem realizing that it is an important area that needs to be addressed. The majority of this research has treated parts as rigid bodies and ignored the flexibility of the parts. Two other researchers have looked at tolerancing flexible assemblies. Gordis [6] applied a frequency domain solution to solving the problem of assemblies with tolerances on bolt holes. Hu [7][8][9] has looked at several different problems involving simple beams, weldments, and sheet metal joints. He developed a method of influence coefficients to calculate the residual stresses in the parts of an assembly. These researchers have laid the groundwork for an important new area of tolerance analysis.

## **ADCATS, AutoCATS, and TI/TOL:**

In 1986 Dr. Ken Chase of Brigham Young University formed a consortium of companies interested in developing computer-aided tolerancing software. This consortium, ADCATS, grew to 12 companies over the years and spawned many new research concepts in tolerancing research.

A fundamental concept is the Direct Linearization Method (DLM). This method takes into account dimensional variation, geometric variation, and kinematic variation (small adjustments) in an assembly of parts. DLM uses a linear approximation of the non-linear assembly functions. This method provides rapid solution of complex problems and easy design iteration on these problems. Gao [10] has shown that the error in the linear approximation of this method is less than 1% for a wide range of assemblies. Another advantage of DLM is that it frames the tolerance analysis problem in terms that are familiar to engineers, such as, datums, vectors, and kinematic joints.

The DLM tolerance modeler has been integrated into several different CAD packages over the years. Currently, there are two versions that are available commercially. AutoCATS is a 2-D version of the software that is integrated into AutoCAD®. This version is available by contacting Dr. Ken Chase at BYU. The other commercial version is linked into Pro/Engineer®. This version is being marketed by Texas Instruments under the name TI/TOL®. The current version of this code is a full 3-D tolerance analysis package. It is integrated with the parametric capabilities of Pro/Engineer®. Both of these packages allow the user to predict the geometric effects of tolerance on an assembly. They will allow the user to estimate the number of rejects in a set of assemblies and indicate which component dimensions are critical in the assembly. They can also indicate areas where gaps or interferences may occur in the assembly and give a statistical variation for these values.

While these tools are useful for dealing with the purely geometric quantities, they do not take into account the inherent flexibility of parts of an assembly. This problem becomes especially apparent in dealing with thin shell components. An airplane skin may be slightly warped and yet it can still be riveted in place. What are the consequences of this? What are the residual stresses created in the part? This question is especially important for assemblies of composite parts. How likely are composite parts to fail during assembly if they are subject to specified tolerance variations?

## **Linear Contact Analysis:**

In 1975 Francavilla and Zienkiewicz[11] published a note documenting a method of calculating contact stresses in parts such as press fit cylinders using the finite element method. This method was limited because it required contact along every point of the mating surfaces. It also ignored the effects of friction. Due to these limitations the method could not handle a wide variety of contact problems that are classified as “moving boundary value” problems. Research in the area of contact

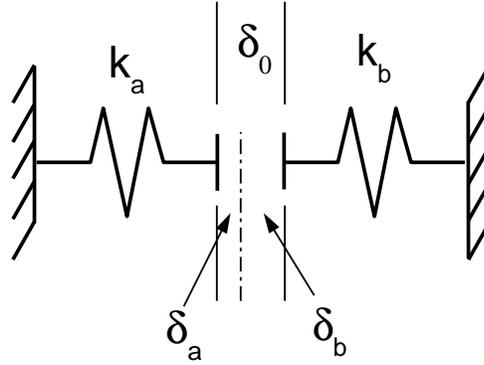


Figure 1: Two springs to be assembled and the resultant equilibrium point.

analysis has ignored this linear method and gone on to develop general contact elements that require nonlinear solution methods. Dealing correctly with these nonlinear elements and solutions is not a trivial problem. The nonlinearities can cause convergence problems and will increase both modeling and solution time.

However, this method is well suited to performing assembly solutions for tolerance analysis. It is also well matched with the DLM tolerance model that is currently used in TI/TOL and AutoCats. The linear contact solution method requires the following assumptions:

1. Contact between mating surfaces must be enforced at the nodes.
2. Small geometric variations in a part will create negligible changes in the part stiffness.
3. Friction must be negligible.
4. The gap/interference must be governed by small deformation theory.
5. The material must behave linearly.
6. The assembly process must be linear.

These limitations are compatible with tolerance analysis, which deals with small derivations about nominal dimensions.

A brief derivation of a method will be described, which combines a rigid body assembly model with a finite element model to predict the effects of tolerance stack-ups on assembly forces and stresses. Let us assume that we have two parts **a** and **b** that are to be brought into contact by closing an assembly gap or interference as shown in figure 1. The gap,  $\delta_0$  can be specified as the sum of the displacements of the individual parts from the equilibrium point

$$\delta_0 = \delta_a + \delta_b. \quad (1)$$

Also, for each part we can write Hooke's Law,

$$\delta_a = F_a/K_a, \quad (2)$$

$$\delta_b = F_b/K_b \quad (3)$$

When the gap is closed, the forces in each spring must be in equilibrium.

$$F_a = F_b = F_{eq}. \quad (4)$$

Combining these equations  $\delta_0$  can be defined as

$$\delta_0 = \left( \frac{1}{K_a} + \frac{1}{K_b} \right) F_{eq} \quad (5)$$

But, since the gap is the known quantity

$$F_{eq} = \frac{1}{\frac{1}{K_a} + \frac{1}{K_b}} \delta_0. \quad (6)$$

In stress analysis, the primary variable of interest is displacement. Substituting equations 4 and 6 into equations 2 and 3 we can obtain the displacements of the individual components

$$\delta_a = \frac{\frac{1}{K_a}}{\frac{1}{K_a} + \frac{1}{K_b}} \delta_0, \quad (7)$$

$$\delta_b = \frac{\frac{1}{K_b}}{\frac{1}{K_a} + \frac{1}{K_b}} \delta_0, \quad (8)$$

The displacements of  $\delta_a$  and  $\delta_b$  depend on the stiffnesses of the mating parts. These results can also be extended to a matrix formulation

$$\{\delta_a\} = \mathbf{K}_a^{-1} [\mathbf{K}_a^{-1} + \mathbf{K}_b^{-1}]^{-1} \{\delta_0\}, \quad (9)$$

$$\{\delta_b\} = \mathbf{K}_b^{-1} [\mathbf{K}_a^{-1} + \mathbf{K}_b^{-1}]^{-1} \{\delta_0\}, \quad (10)$$

where  $\{\delta_a\}$ ,  $\{\delta_b\}$  and  $\{\delta_0\}$  are displacement vectors and  $\mathbf{K}_a$  and  $\mathbf{K}_b$  are stiffness matrices. This information can be used to design bolted or bonded joints in assemblies.

This process involves three matrix inversions so it is not numerically cheap. However, it does not require any iterations and there are no convergence problems. Because of the assumption of linearity, many different cases of  $\{\delta_0\}$  may be evaluated without recalculating the stiffness matrices.

As was noted previously, this method requires three matrix inversions, so we would like to find a way to deal with smaller matrices. Also, the stiffness matrices are likely to be different sizes which poses problems with the matrix multiplications described in equations 9 and 10. Super-elements can be used to alleviate both problems. The super-element describes the equivalent stiffness for an entire part in terms of the degrees of freedom along its boundary. If we use only the contact nodes to define the super-element boundary, and equivalence the nodes along the contact boundary the parts may be assembled using simple matrix algebra. Also, the matrix size is reduced to the number of boundary nodes times the number of degrees of freedom at each node. It should be noted that while the total matrix size is reduced, the super-element stiffness matrix is not sparse and the process of forming the super-element requires an additional inversion.

The following sections will provide examples of this technique.

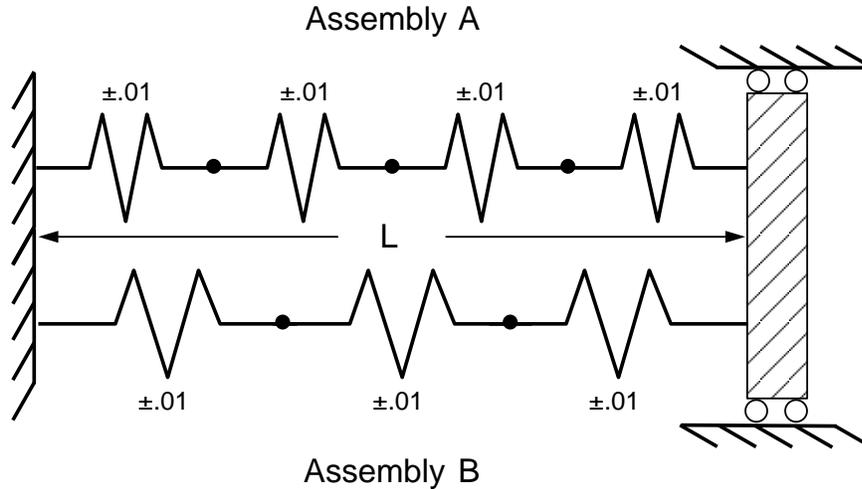


Figure 2: An assembly of linear springs.

### Example 1: One dimensional springs.

The first example is a simple one-dimensional series spring problem shown in figure 2. This problem is formulated such that all the mathematics may be easily solved by hand.

Consider a linear assembly of springs with two sub-assemblies A and B. The spring lengths are subject to random manufacturing variations. Each spring in the sub-assembly A has an unloaded length of  $L/4 \pm .01$  and each spring in sub-assembly B has an unloaded length of  $L/3 \pm .01$ . We will assume that each tolerance  $T$  represent a  $3\sigma$  (standard deviation) variation in the length. In this simple one-dimensional problem, the component tolerances may be added by a root-sum-squares technique to obtain a statistical estimate of the variation in each assembly. Thus, the statistical assembly variation due to the accumulated component tolerances is:

$$\delta_{asm} = \sqrt{\sum T_i^2} \quad (11)$$

For assembly A this result is:

$$\delta_a = \sqrt{4(.01)^2} = \pm .02 \quad (12)$$

For assembly B:

$$\delta_b = \sqrt{3(.01)^2} = \pm .0173 \quad (13)$$

The variation of the gap between the free length of these two springs assemblies can be defined by the root-sum-squares of the variation of each assembly.

$$\delta_0 = \sqrt{\delta_a^2 + \delta_b^2} = \sqrt{4(.01)^2 + 3(.01)^2} = \pm .0265 \quad (14)$$

If the nominal lengths of A and B are equal, this value can be visualized as the maximum probable difference between the ends of the springs due to manufactur-

ing errors. If the nominal lengths are *not* the same, then there is an average gap which must be accounted for as well.

If the two sets of springs are assembled as shown in figure 2, one will be stretched and the other compressed until they are the same length. This deformation will result in a force in the springs that will depend on the size of the gap. The equilibrium position of the combined assembly must be a function of the stiffnesses of the springs.

If the springs are assumed to act as simple trusses, the stiffnesses matrices for each assembly can be defined as

$$K_a = \frac{EA}{L_a} \begin{bmatrix} 1 & -1 & 0 & 0 & 0 \\ -1 & 2 & -1 & 0 & 0 \\ 0 & -1 & 2 & -1 & 0 \\ 0 & 0 & -1 & 2 & -1 \\ 0 & 0 & 0 & -1 & 1 \end{bmatrix} \quad (15)$$

and

$$K_b = \frac{EA_2}{L_b} \begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}. \quad (16)$$

If the matrices are reduced via matrix condensation techniques (super-elements) they become

$$K_a = \frac{EA}{L_a} \begin{bmatrix} \frac{1}{4} & -\frac{1}{4} \\ -\frac{1}{4} & \frac{1}{4} \end{bmatrix} \quad (17)$$

and

$$K_b = \frac{EA_2}{L_b} \begin{bmatrix} \frac{1}{3} & -\frac{1}{3} \\ -\frac{1}{3} & \frac{1}{3} \end{bmatrix} \quad (18)$$

When the ground nodes are constrained, this is reduced even further to

$$K_a = \frac{EA}{4L_a} \quad \text{and} \quad K_b = \frac{EA_2}{3L_b}. \quad (19)$$

These results may be substituted into equations 9 and 10

$$\delta_a = \frac{K_a^{-1}}{K_a^{-1} + K_b^{-1}} \delta_0 = \frac{4L_a A_2}{4L_a A_2 + 3L_b A} \delta_0 \quad (20)$$

$$\delta_b = \frac{K_b^{-1}}{K_a^{-1} + K_b^{-1}} \delta_0 = \frac{3L_b A}{4L_a A_2 + 3L_b A} \delta_0 \quad (21)$$

but,  $L_a = L/4$  and  $L_b = L/3$ . Also let  $A_2 = 2A$ , so

$$\delta_a = \frac{2}{3} \delta_0 \quad \text{and} \quad \delta_b = \frac{1}{3} \delta_0 \quad (22)$$

Assembly **A** has more components and a larger variation than assembly **B**. The resulting deformations represent a statistical variation in the displacement of the springs in each sub-assembly.  $\delta_a$  and  $\delta_b$  become boundary conditions applied to the gap surfaces.

The assembly force required to close the gap is proportional to the statistical displacement of the springs. The nominal force in each sub-assembly of springs is zero when the ideal length of each subset is equal to  $L$ . However, accounting for the  $3\sigma$  variation in the lengths induces an equilibrating force of

$$F_{eq} = K_a \delta_a = K_b \delta_b \quad (23)$$

$$F_{eq} = \pm .0176 \frac{EA}{L} \quad (24)$$

in each sub-assembly.

These stresses can be significant. If we assume steel trusses with a 10:1 ratio of length to area, the equilibrium force in each sub-assembly is 53,900 lbs.

Thermal stress calculations are frequently performed because it has been shown that these stresses can be significant. Yet the displacements due to manufacturing variation are on the same order of magnitude as thermal expansion. For example, assume that we have a steel part that is subjected to 100°F temperature rise.

$$\begin{aligned} \alpha_{steel} &= 6 \times 10^{-6} \\ \sigma &= E\alpha\Delta T \\ &= (30 \times 10^6)(6 \times 10^{-6})100 \\ &= 18,000\text{psi}. \end{aligned} \quad (25)$$

This example shows that the thermal stresses and the manufacturing stresses can be of the same order of magnitude.

In the following examples, the finite element model and super-elements were defined with MSC/PATRAN. The super-element stiffness matrices were assembled using MSC/NASTRAN and written to a file in that could be read by MATLAB. The necessary matrix manipulations are easily obtained in MATLAB. We will also demonstrate how to solve this problem directly in MSC/NASTRAN using multi-point constraints.

## Example 2: Simple block assembly

Figure 3 shows three blocks that fit inside a base. When the parts are considered as rigid bodies, it can be considered as a one-dimensional tolerance stack-up; however, when deformations are considered, it becomes a two-dimensional problem. The nominal dimensions on the assembly are specified such that the blocks are press fit into the base with a nominal interference of .025 inches. The tolerance on this interference is specified to be no more than  $\pm .025$  inches. Summing the

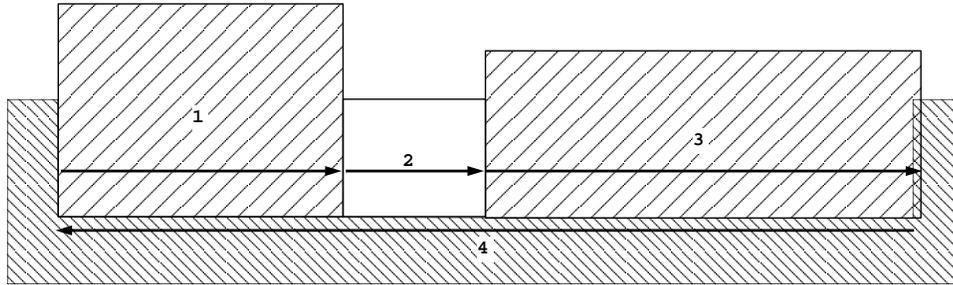


Figure 3: One-dimensional stacked block problem with an interference fit.

nominal and variable interference gives a range from 0 to .05 inches. The parts are aluminum with a yield strength of 70 ksi. They are to be assembled without yielding the material. We desire to know how many parts will be rejected with the given design specifications.

The finite element model is constrained in the X and Y directions at the midpoint of the base. The rest of the base is on rollers. There are also rollers between the blocks and base to allow kinematic adjustments. (This constraint could also have been defined as a sliding plane MPC.) We will assume that the frictional effects are very high between the mating surfaces on the left and right edges so that the nodes on the vertical edges will move together. The blocks are assumed to be thick relative to their length and width, so this problem can be treated as plane strain.

The model is constructed so that the blocks fit exactly inside the base. The nominal interference will be accounted for in the displacement boundary condition applied at the gap. This simplifies the construction process and corresponds with the assumption that small variations in geometry will create negligible changes in the stiffness of the parts.

This problem was formulated in MSC/PATRAN. The following is the procedure that was used to create and analyze the model.

1. Each component was placed in a super-element.
2. MSC/NASTRAN was used to create the super-element stiffness matrices and a DMAP was written to print the stiffness matrices into the output file.
3. The stiffness matrices were extracted from the data file and manipulated using MATLAB to perform the matrix operations described earlier.
4. The maximum interference of .05 inches was imposed at the gap.
5. The results of this analysis were then imported back into MSC/PATRAN for post-processing.

Figures 4 and 5 show the results. The base has expanded horizontally with the greatest deflection occurring in the side rails. The blocks have contracted horizontally with the greatest contraction occurring near the base. However, the peak stress is reported at 102 ksi., which exceeds the design limit of 70 ksi. In

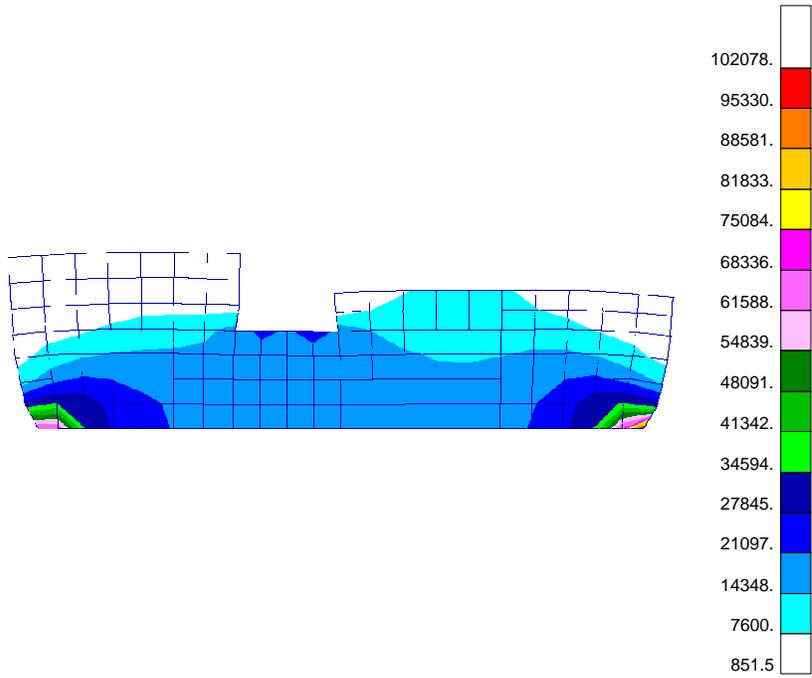


Figure 4: Deflection in blocks due to interference fit.

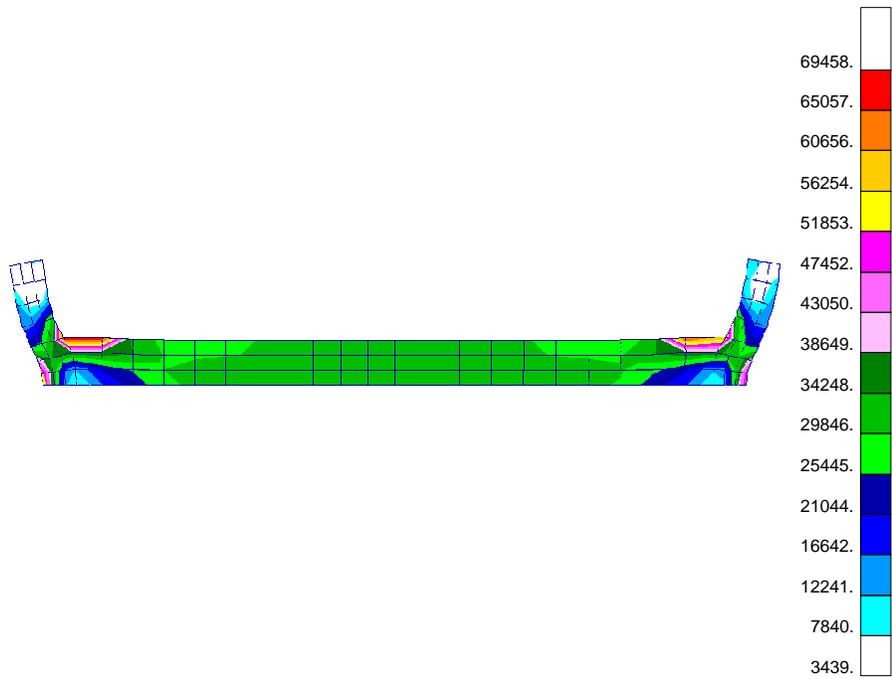


Figure 5: Deflection in base due to interference fit.

fact, the material has exceeded the plastic limit and will never really reach 102 ksi. This analysis can only be used to describe the deformations that will be in the linear range. Since the results are linear, a simple scale factor, (70/120) may be applied to the maximum interference to reduce the max stress. All the stresses and displacements throughout the assembly will be reduced by the same fraction.

If a normal distribution is assumed for the interference and a linear model is assumed for the stress/strain relationship, a simple scale factor can also be defined for the statistical stress space. As the interference varies from 0 to .05 inches the maximum Von Mises stress in the assembly varies from 0 to 102 ksi. The mean Von Mises stress must be 51 ksi corresponding to the mean interference of .025 inches.. This distribution is shown in figure 6. Above 70 ksi, the maximum stress and the distribution of this stress is unknown from the linear analysis. There will be a discontinuity at the yield limit and it will continue with a different distribution.

If the stress distribution about the mean is assumed to be a  $3\sigma$  variation, the number of standard deviations corresponding to 70 ksi can be calculated by

$$x = \frac{70 - 51}{(51 - 0)/3} = 1.12. \tag{26}$$

By using a standard statistical table for normal distributions, this value corresponds to 86.86%. This means that 13.14% of all assemblies will exceed the design limits and be rejected due to improperly assigned tolerances.

Note that even though the material properties extended into the plastic region, only the linear results were used to predict the number of parts that exceeded the yield strength. A non-linear analysis was not required. This type of design constraint is a common occurrence where members such as bolts are taken just to the yielding point or where composite members are functional while the interlaminar shear does not induce delamination.

Similar results can be obtained using carefully defined multi-point constraint (MPC) equations. The MPC model is shown in figure 7. The nodes on the blocks are related to the nodes on the base, but there is one more node that is added into

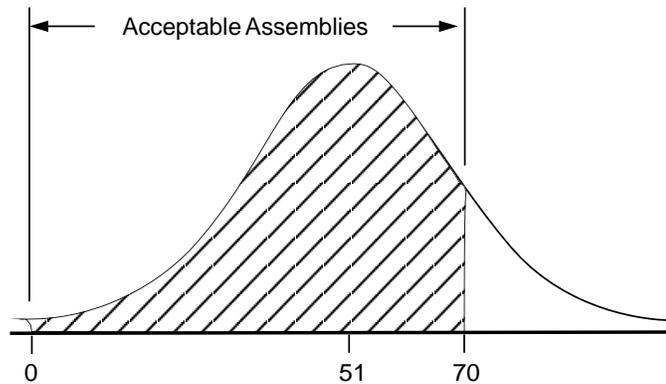


Figure 6: Statistical distribution of maximum Von Mises Stress.

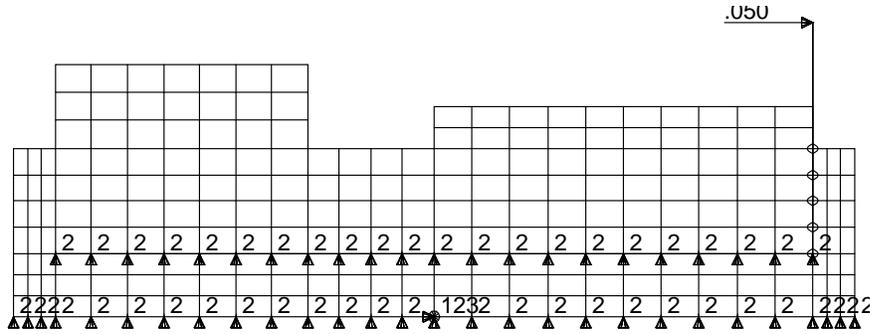


Figure 7: Model defined using MPC's.

the equation. The node at the top of figure 7 is used to store the gap or interference data. An enforced displacement constraint of .05 inches has been placed on this node. A sample MPC input line for MSC/NASTRAN would appear as:

```
MPC,5,1,1,-1.0,1000,1,1.0
, ,501,1,1.0
```

This MPC relates the X displacements of node 1 (on the right block) to node 1000 (the node storing the gap information) and node 501 (on the base). It enforces the interference and provides the equilibrium solution that accounts for model stiffness. A similar MPC must be defined for every point of contact. The results from this method are compared with super-element stiffness matrices in table 1.

These results show that both methods give comparable results. The difference in this analysis is possibly due to the coupling of the Y components (via Poisson's ratio) in the stiffness matrices that was not enforced in the MPC equations.

Table 1: A Comparison of Stiffness Matrix Manipulation vs. MPC's

Node Number	X Displacement with MPC's	X Displacement with Stiffness Matrices	Percent Error
1	0.0212	0.0212	0.1641
2	0.0204	0.0203	0.6665
3	0.0192	0.0191	0.6611
4	0.0175	0.0175	0.1395
5	0.0142	0.0143	0.4437
6	-0.0209	-0.0211	0.9689
7	-0.0200	-0.0202	0.7092
8	-0.0189	-0.0190	0.4728
9	-0.0175	-0.0175	0.1482
10	-0.0142	-0.0142	0.0128

Multi-point constraints are fairly easy to define and variable gaps/interferences can be defined by using multiple “gap nodes”. However, they do not provide access to the sensitivity terms that can be used to determine the effect of variations at each node. The sensitivity terms are directly available from the stiffness matrices as derived in equations 9 and 10. This information is useful in allocating tolerances based on design results. Both of these analysis techniques are useful depending upon the desired results.

## **Conclusion:**

This paper has presented a method for combining tolerance analysis with model flexibility to show the consequences of manufacturing variations on assembly stresses. Only one-dimensional assemblies were examined to demonstrate the technique and the importance of this new technology. This technique is being verified on real world problems with multiple element types and both two and three-dimensional tolerance variation.

This tool is useful for a wide variety of design and manufacturing tasks:

- Predicting the final location of mating surfaces.
- Predicting distortion due to internal assembly stresses.
- Predicting internal stress and force due to assembly of off-nominal geometry parts.
- Predicting percent of assemblies which will not meet design limits.
- Performing “what-if” studies and assigning tolerances throughout an assembly to minimize production/maintenance problems.
- Performing sensitivity studies to identify the critical sources of variation.

This analysis method helps engineers and designers understand the effects and the importance of manufacturing tolerance early in the design process. It provides a tool for evaluating the consequences of manufacturing tolerances on assembled products. It can serve as a design tool by using estimated process variation to assign tolerances. It can also be used with actual process data to determine the affects of manufacturing variation on assemblies.

## **Acknowledments**

This work was sponsored by the Office of Naval Research, contract #N00014-92-J-4064.

## References

- [1] *MSC/PATRAN User's Manual*, The MacNeal-Schwendler Corp., Los Angeles, CA, 1995.
- [2] *MSC/NASTRAN User's Manual Version 68*, The MacNeal-Schwendler Corp., Los Angeles, CA, 1995.
- [3] R. Hillyard and I. Braid, "Analysis of dimensions and tolerances in computer-aided mechanical design," *Computer-Aided Design*, vol. 10, pp. 161–166, May 1978.
- [4] A. A. G. Requicha, "Representation of tolerance in solid modeling: Issues and alternative approaches," in *Solid Modeling by Computers: from Theory to Application* (J. Boyse and M. Pickett, eds.), pp. 3–22, New York: Plenum Press, 1984.
- [5] A. A. G. Requicha and S. C. Chan, "Representation of geometric features, tolerances, and attributes in solid modelers based on constructive geometry," *IEEE Journal of Robotics and Automation*, vol. RA-2, Sept. 1986.
- [6] J. Gordis and W. Flannelly, "Analysis of stress due to fastener tolerance in assembled components," *AIAA Journal*, vol. 32, pp. 2440–2446, Dec. 1994.
- [7] S. Liu and S. Hu, "Variation simulation for deformable sheet metal assemblies using finite element methods," *ASME Journal of Engineering for Industry*, preprint 1995.
- [8] S. Liu, S. Hu, and T. Woo, "Tolerance analysis for sheet metal assemblies," *ASME Journal of Mechanical Design*, preprint 1995.
- [9] S. Liu, H.-W. Lee, and S. Hu, "Variation simulation for deformable sheet metal assemblies using mechanistic models," *Transactions of the North American Manufacturing Research Institute of SME*, May 1995.
- [10] J. Gao, *Nonlinear Tolerance Analysis of Mechanical Assemblies*. PhD thesis, Brigham Young University, Provo, UT, Aug. 1993.
- [11] A. Francavilla and O. Zienkiewicz, "A note on numerical computation of elastic contact problems: surface contact and impact," *International Journal for Numerical Methods in Engineering*, vol. 9, pp. 913–924, 1975.