HEAVY MOVABLE STRUCTURES, INC.
THIRTEENTH BIENNIAL SYMPOSIUM

October 25-28, 2010

ANSYS Non-Linear Multi staging Technique for Shrink Fit Analysis

Dr. Yahia M. Al-Smadi, PhD
Dr. Yousef Alostaz, PhD, PE
Dr. Hamid Adib, PhD, PE, F.ASCE

AECOM

CARIBE ROYAL HOTEL
ORLANDO, FLORIDA

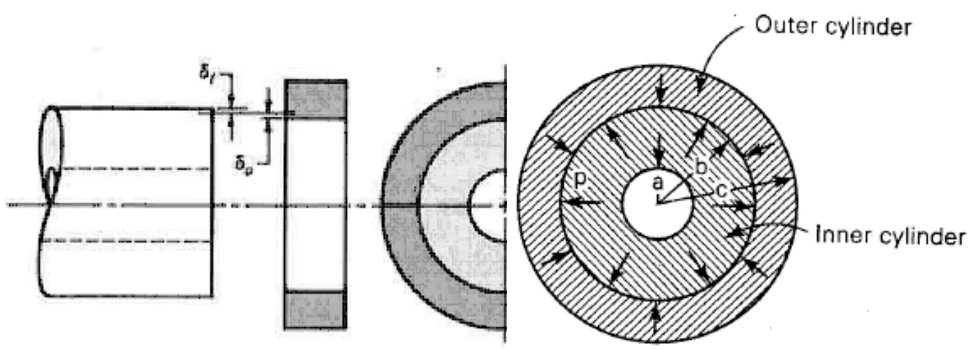
Abstract

Shrink Fit procedure is widely used in industry and could be very complicated to analyze due to the irregularities of the fitted parts. This complexity comes from many factors including; geometry, materials, loadings, number of fitted parts, or any combination of these factors. The use of conventional analysis equations could help as a benchmark to estimate stress and deflection but will not be as more descriptive as using a finite element method (FEM). In previous work [1], the author showed the implication of using (FEM) to solve for geometrical complexities. This paper focuses on the consequences of using FEM to solve for non-linear analysis and introduce a new technique to analyze the shrink fit for many fitted parts. The results of analyses presented herein will show that design and analyses of any complex shrink fit situation can be achieved through the use of FEM with consideration of non-linear analysis and multistage technique.

A reliable and robust analysis software (ANSYS) was used to complete both 3D FE modeling and analysis. Analysis model parameters such as material properties, non-linear behavior of the material, and meshing are presented and discussed.

Introduction

Conventional shrink fit analysis is a well-established field where the objective function is to calculate the shrink fit pressure, other stresses and deformation. This analysis is based primarily on axisymetrical condition of the engaged parts. In other words, fitted parts shall have constant cross section and material during the shrink fit area as shown in Figure 1a.



a) b)

Figure 1: schematic diagram for shrink fit a) Before Press Fit b) After Press Fit

An example of many parts fitted together would be trunnion girders with trunnion shafts can be found in small and medium bascule bridges. Extensive nonlinear study about a trunnion girder was presented in previous work [1]. In this paper, emphasis will be given to the non-linear analysis and the effect of multistage technique in shrink fit analysis. In the next sections, the author will present an FEM model for shrink fit scenarios that focus on the scope of this paper.

Conventional Shrink Fit Method

Shrink fitting is encountered in many engineering designs. It refers to fitting an object into a slightly smaller cavity. Due to normal forces to the surface that develop at the interface, the inner object shrinks while the outer object expands. The amount of shrinkage or expansion is determined by the material properties as well as the geometry of the components. This fit generates a pressure determined by equations 1 and 2. The radial and tangential stresses are determined by equations 3and 4. The radial deflection in any part after the assembly shown in equation 5, Von Misses Stress and shear stress are shown in equations 6 and 7. The required equations for the analysis, in their general form, are shown below.

$$p_{Shrink\_Fit} := \frac{\delta}{2 \cdot b \cdot \left[\frac{1}{E_o} \cdot \left(\frac{c^2 + b^2}{c^2 - b^2} + \nu_o \right) + \frac{1}{E_i} \cdot \left(\frac{b^2 + a^2}{b^2 - a^2} + \nu_i \right) \right]} \quad \text{for different materials} \qquad \text{Eqn 1}$$

$$p_{Shrink\_Fit} := \frac{E \cdot \delta}{b} \cdot \frac{(b^2 - a^2)(c^2 - b^2)}{2 \cdot b^2 (c^2 - a^2)} \quad \text{for similar materials} \qquad \text{Eqn 2}$$

$$\sigma_r(r) := \frac{a^2 \cdot p_i - b^2 \cdot p_o}{b^2 - a^2} - \frac{(p_i - p_o) \cdot a^2 \cdot b^2}{b^2 - a^2} \cdot \frac{1}{r^2} \qquad \text{Eqn 3}$$

$$\sigma_t(r) := \frac{a^2 \cdot p_i - b^2 \cdot p_o}{b^2 - a^2} + \frac{(p_i - p_o) \cdot a^2 \cdot b^2}{(b^2 - a^2) \cdot r^2} \qquad \text{Eqn 4}$$

$$u(r) := \frac{1 - \nu}{E} \cdot \frac{(a^2 \cdot p_i - b^2 \cdot p_o) \cdot r}{b^2 - a^2} + \frac{1 + \nu}{E} \cdot \frac{(p_i - p_o) \cdot a^2 \cdot b^2}{(b^2 - a^2) \cdot r} \qquad \text{Eqn 5}$$

$$\sigma_{Von\_Mises}(r) := \sqrt{\sigma_t(r)^2 + \sigma_r(r)^2 - \sigma_t(r) \cdot \sigma_r(r)} \qquad \text{Eqn 6}$$

$$\tau_{max} := \frac{1}{2} \cdot (\sigma_t - \sigma_r) \qquad \text{Eqn 7}$$

Where

p_{Shrink_Fit} : Pressure due to shrink fit at radius b as shown in Figure 1
E : Modulus of Elasticity of the steel 29×10^6 psi
δ : Interference fit
a : inner radius of the shaft as shown in Figure 1
b : outer radius of the shaft and inner radius of the cylinder as shown in Figure 1
c : outer radius of the cylinder as shown in Figure 1
r : arbitrary distance between a and b or between b and c
$\sigma_r(r)$: radial stress in the shaft as function of r. For the cylinder replace a and b with b and c
$\sigma_t(r)$: tangential stress in the shaft as function of r. For the cylinder replace a and b with b and c
u(r) : radial displacement in the shaft at any arbitrary distance.
ν: Poisson ratio 0.29 for steel
τ_{max} : maximum shear stress

Description of the Model

The model consists of 4 main parts as shown in Figure 2; shaft, bushing, gear, and collar. Schematic diagram for part dimensions is shown in Figure 3. All parts were considered as structural steel ASTM Gr.50 with 50ksi yield strength. The main focus is to show the importance of considering nonlinear analysis and multistage technique. More discussion about material and analysis assumption is presented in later sections.

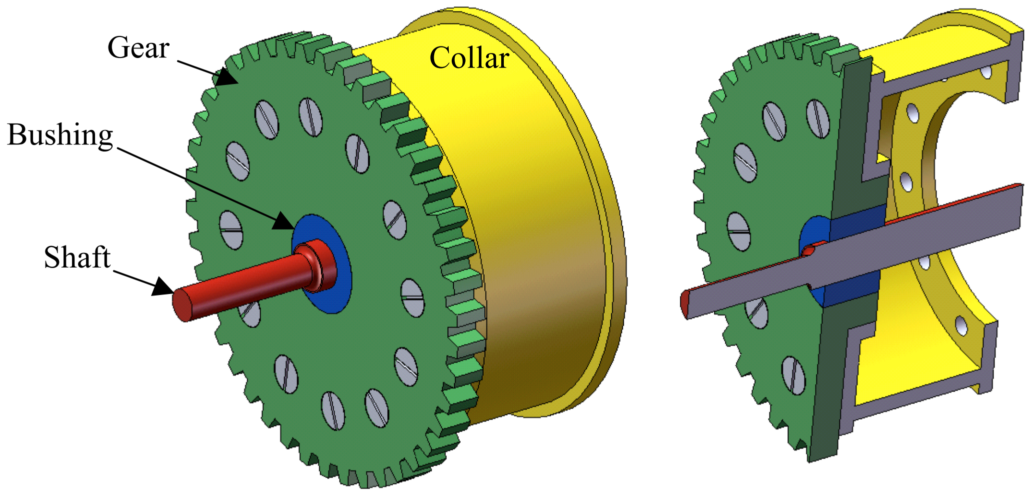
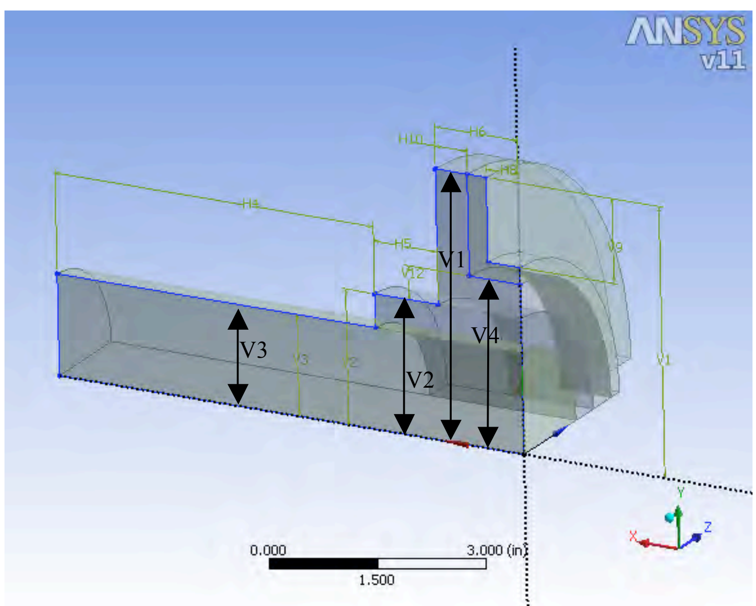


Figure 2: 3D model used in ANSYS analysis



Where;

V1 = 4 inch
V2 = 2 inch
V3 = 1.5 inch
V4 = 2.5 inch
δ_{V3} = 0.001 inch
δ_{V2} = 0.006 inch
δ_{V4} = 0.006 inch

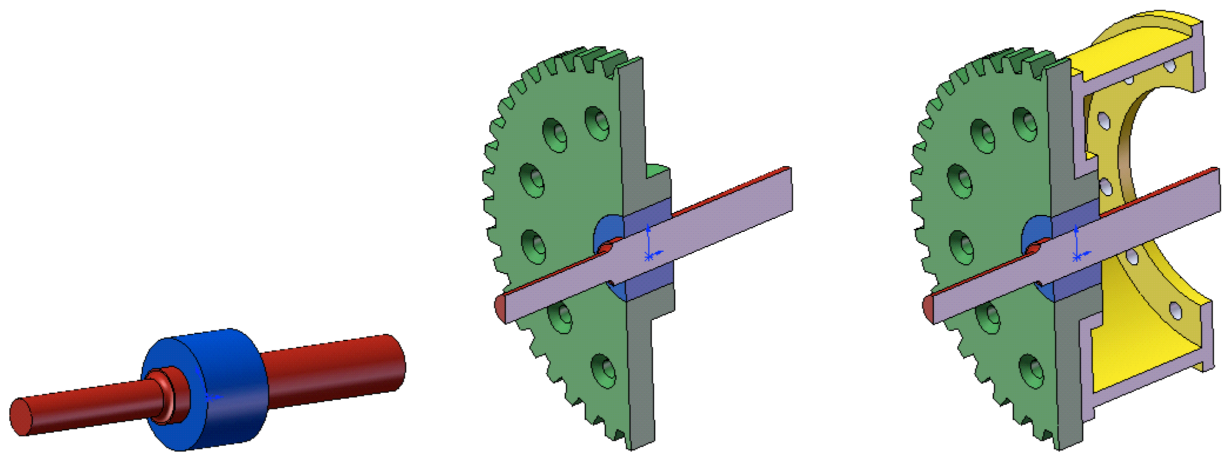
Figure 3: Schematic diagram for fitted parts

Staged Analysis Approach

The 3-D models were analyzed with three installation stages. Staging technique is performed by activating or reactivating the elements for the assigned stage. Stage stresses and deflections are cumulatively added throughout the different stages. The following stages were considered based on the presented shrink fit model.

1. Stage 1: Shaft and bushing.
2. Stage 2: Shaft, bushing, and gear.
3. Stage 3: Shaft, bushing, gear, and collar.

Figure 4 shows the 3-D models for each stage

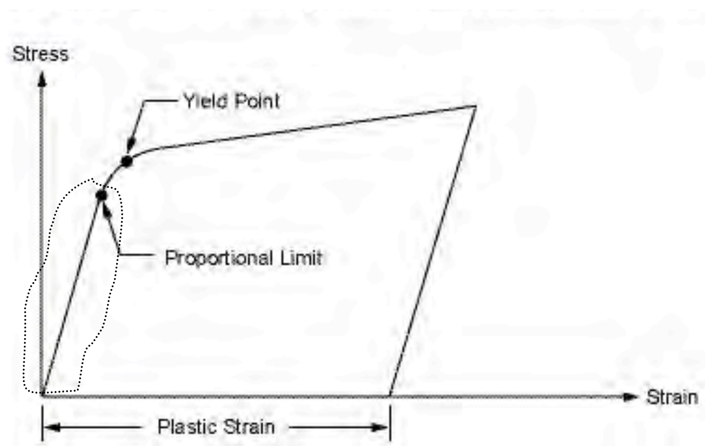


Stage 1 Stage 2 Stage3

Figure 4: 3D models represent the stages used in ANSYS multistage analysis

Description of the Material Models

The material used in this analysis is structural steel which considered as linear if the design stresses don't exceed design limit (proportional limit in **Error! Reference source not found.**). This material reveals a linear stress-strain relationship up to a yield point as shown in **Error! Reference source not found.**. Beyond this limit, the stress-strain relationship will become nonlinear. Plastic behavior begins when stresses exceed the material's yield point. In this paper, a comparison is made between linear versus non-linear material.



A problem becomes non-linear if the loading causes significant changes in the structure stiffness or stress levels approach the yield point beyond the elastic limit (plasticity) or large displacements occur.

In order to capture the plastic response, load is to be applied in a sequence of small incremental load steps (multistep loading) or small time steps, which will affect the solution if large deformation is expected to occur.

Structural Steel is generally modeled in ANSYS as elastic isotropic material. The definition for the non-linear model is Bilinear Isotropic Hardening shown in Figure 5 where the yield stress is set at 50ksi and the plastic hardening continues to 55ksi stress. Below is a description of non-linear models as defined by ANSYS:

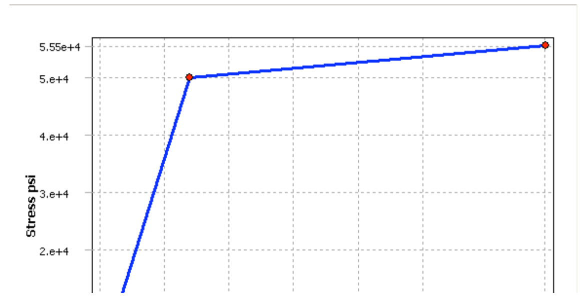
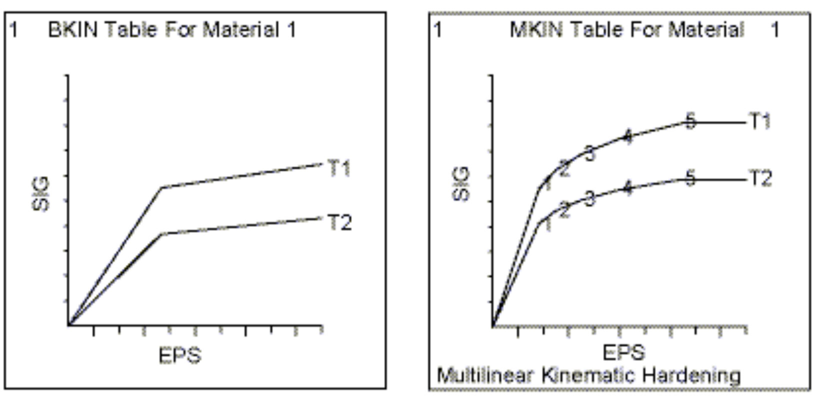


Figure 5: Bilinear Isotropic hardening curve for structural steel

1. **Bilinear Isotropic Hardening** (BISO) option uses the von Mises yield criteria coupled with an isotropic work hardening assumption. This option is often preferred for large strain analyses.
2. **Multilinear Isotropic Hardening** (MISO) option is like the bilinear isotropic hardening option, except that a multilinear curve is used instead of a bilinear curve. It is, however, recommended for large strain analyses. The MISO contains the hyperbolic stress-strain relationship developed by Filz, et al. (1990).
3. **Bilinear Kinematic Hardening** (BKIN) This option as shown in Figure 6 is recommended for general small-strain use for materials that obey von Mises yield criteria (which includes most metals). It is not recommended for large-strain applications.
4. **Multilinear Kinematic Hardening** (MKIN) This option as shown in Figure 6 uses multiple curves to describe strss-strain relationship for materials with temperature properties, each curve should contain the same number of points. The assumption is that the corresponding points on the different stress-strain curves represent the temperature dependent yield behavior of a particular sublayer. These options are not recommended for large-strain analyses.



a) b)

Figure 6: Kinematic Hardening (a) Bilinear kinematic hardening, (b) Multilinear kinematic hardening

Modeling and Analysis Assumption

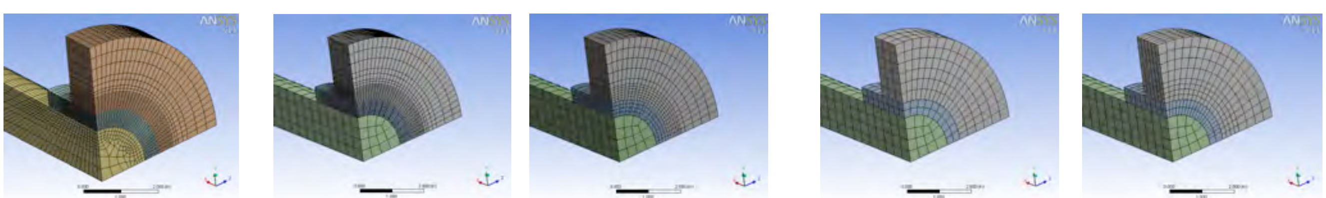
- No bolts and bolt holes
- No filets
- No gear teeth
- No collar body, only the shrink fitted collar hub
- ¼ model was used with symmetry constrain on the sectioned planed.
- Material used for all parts is ASTM A709 Gr50

Analysis Results

The analysis was performed to ensure adequate load cases and scenarios achieving the paper objectives. Fifteen models were investigated, the first fives models performed for the purposes of sensitivity study, in which the mesh refinement of the contact surface between fitted parts was decreasing, the overall mesh size for the parts was set to default ANSYS values.
The remaining models use one mesh size for the contact elements with a resolution of 0.1inch. Each model was classified based on two criteria; linear or non-linear analysis and staged or non-staged. Table 1 shows the specifications of each model; notices that the elapsed time shown in the last column indicates the total completion time for the analysis.

Sensitivity Analysis

The purpose of the sensitivity analysis is to find an acceptable contact mesh size to be used for the model numbers 6-15 in Table 1. The optimized mesh size shall not sacrifice the accuracy of the results and consumes less computational time, Figure 7 shows the meshed models used in the sensitivity analysis. The results of this analysis are shown in the first five rows of Table 1 and Table 2. Table 1 shows that the time required to perform shrink fit increases exponentially with the refinement of the contact mesh size, the results were normalized and shown in Figure 8. To determine the optimized mesh size, the results of hoop stress on the contact surfaces were normalized and shown in Figure 8. It is clearly noticed that hoop stress is almost constant between models 1 and 3 and then increases with less mesh refinement. Comparing hoop stresses to the elapsed time (Figure 8), model 3 with mesh contact size of 0.1inch would be considered as an optimum solution. Model 3 gives hoop stress of 0.1% the less than Model 2 and 0.13% less than Model 1, Model 3 takes 2min to solve compared to 9min for Model 2 and 21min for Model 1. As a consequence, the remaining models were meshed with 0.1 inch and the results are shown in Table 2 and Table 3.



0.03 inch mesh Model 1 | 0.05 inch mesh Model 2 | 0.1 inch mesh Model 3 | 0.2 inch mesh Model 4 | 0.3 inch mesh Model 5

Figure 7: Sensitivity study shows different analyzed models with the assigned mesh

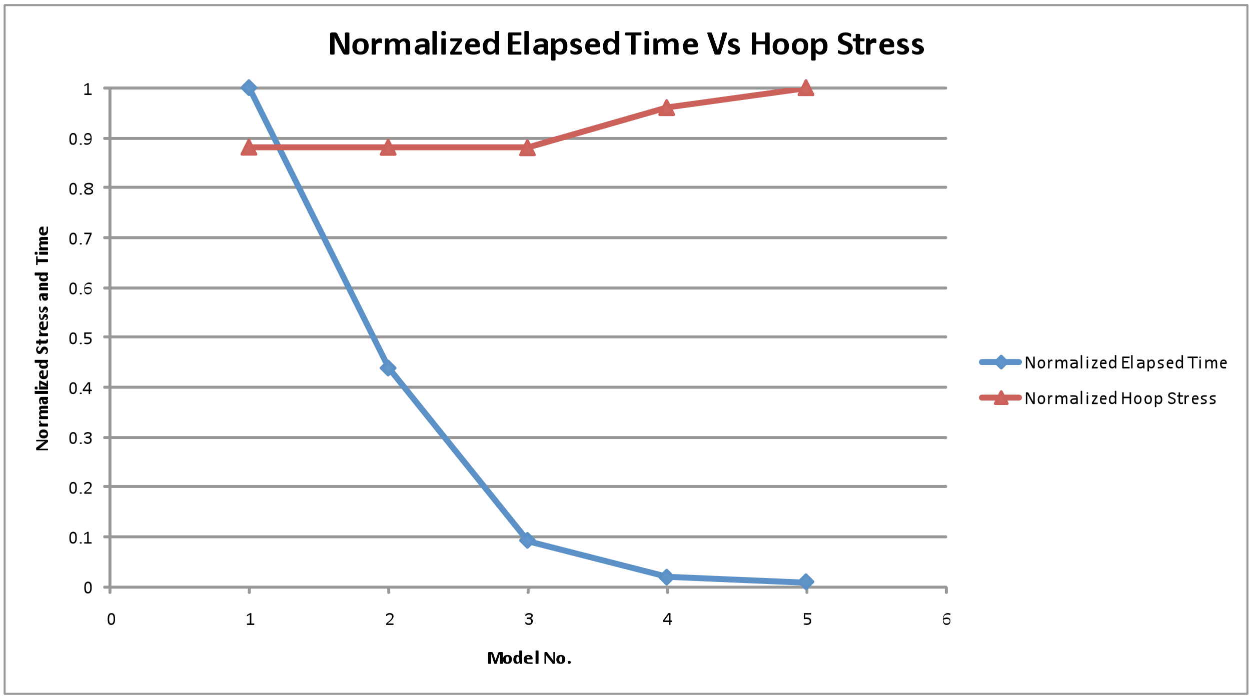


Figure 8: Sensitivity Study shows different meshed models.

Table 1: ANSYS Classification for 3-D Model

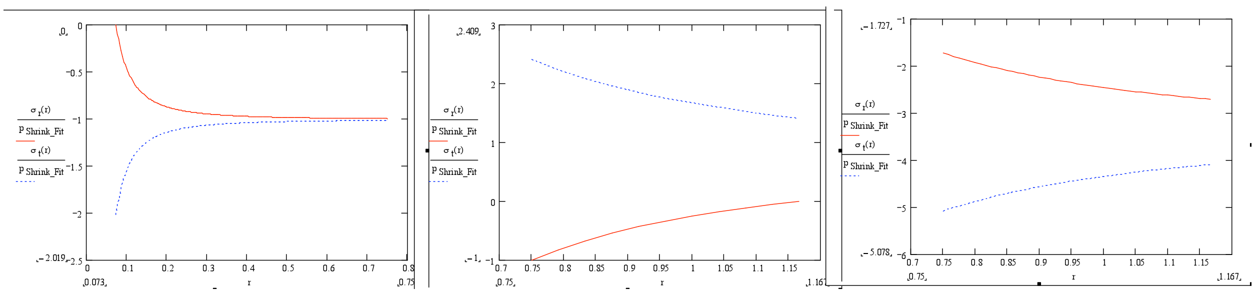
Group #	Model #	No. Parts	Contact Mesh Size (inch)	Linear	Non-Linear	Staged	Non-Staged	Analysis Elapsed Time (min)
1	Model 1	2	0.03		✓		✓	21.7
1	Model 2	2	0.05		✓		✓	9.5
1	Model 3	2	0.1		✓		✓	2
1	Model 4	2	0.2		✓		✓	0.4
1	Model 5	2	0.3		✓		✓	0.2
2	Model 6	2	0.1	✓			✓	0.3
2	Model 7	2	0.1		✓		✓	2
3	Model 8	3	0.1	✓			✓	3
3	Model 9	3	0.1	✓		✓		99
3	Model 10	3	0.1		✓		✓	4
3	Model 11	3	0.1		✓	✓		26
4	Model 12	4	0.1	✓			✓	4
4	Model 13	4	0.1	✓		✓		15
4	Model 14	4	0.1		✓		✓	5
4	Model 15	4	0.1		✓	✓		51

Analyses Result Discussion

Three part and four part models were analyzed using two options; linear and non-linear analysis. Table 1 indicates the classification of Models 8 through 15 using staged and non-staged technique. It also shows the completion time for each performed analysis.

The focus in this paper will be studying model groups 3 and 4. Group 3 includes three part model numbers 8-11 and Group 4 indicates four part model numbers 12-15.

Results of hoop stresses and radial stresses for fitted parts that are subjected to external pressure only such as the shaft follow the distribution illustrated in Figure 9a. Hoop stress (dotted line) and radial stress (solid line). The radial stress is minimum at the inner radius while the maximum magnitude of the circumferential stress (hoop stress) occurs at the inside surface of the bushing.



a) b) c)

Figure 9: distribution of normalized hoop (dotted) and radial (solid) stresses. a) External pressure only, b) internal pressure only, and c) internal and external pressure.

Figure 9b illustrates the distribution of hoop stress (dotted line) and radial stress (solid line) in the fitted parts that are subject to internal pressure only such as the gear in model numbers 8-11 and collar in model numbers 12-15. Both stresses decrease as the radius increase. Radial stress becomes negligible at the outer surface. Figure 9c illustrates the distribution of hoop stress (dotted line) and radial stress (solid line) in the fitted parts that are subject to internal and external pressure such as bushing in model numbers 11-15 and gear in model numbers 12-15.

Note that the analysis for Models 9 and 13 (non-linear with staged analysis) gives unpredictable results. Several attempts were made to analyze these models with different assumptions. However, results were almost close to the result shown in Table 2.

Table 2: ANSYS Hoop Stress Results for 3-D Fitted Assembly

Model #	Shaft		Bushing		Gear		Collar	
	σ_I (ksi)	σ_o (ksi)	σ_I (ksi)	σ_o (ksi)	σ_I (ksi)	σ_o (ksi)	σ_I (ksi)	σ_o (ksi)
Model 1		-31.33			31.27			
Model 2		-31.74			31.24			
Model 3		-30.6			31.18			
Model 4		-28.3			34			
Model 5		-26.7			35.4			
Model 6	-22.5	-30.3			50.5	18.8		
Model 7	-23.3	-30.6			31.1	20.5		
Model 8	-36.1	-41.4	-20.5	-28.8	67.3	25.3		
Model 9	-5.1	-52.4	16	11.2	-0.007	0		
Model 10	-24.5	-28.1	-23.6	-28.5	32.2	20.5		
Model 11	-23.3	-27.4	-22.5	-28.5	30.9	19.5		
Model 12	-46	-48	-27	-35	62	36	111	98
Model 13	-4	-5	16	10	0.004	0	0	0
Model 14	-31	-29	-24	-26	31	36	44	51
Model 15	-24	-25	-19	-21	31	32	49	43

Table 3: ANSYS Radial Deflection Results for 3-D Fitted Assembly

Model #	Shaft		Bushing		Gear		Collar	
	Δ_I (in)	Δ_o (in)	Δ_I (in)	Δ_o (in)	Δ_I (in)	Δ_o (in)	Δ_I (in)	Δ_o (in)
Model 5	-2.4e-05	-1.2e-03			3.8e-03	2.6e-03		
Model 6	-3.0e-05	-1.3e-03			4.2e-03	2.8e-03		
Model 7	-8.2e-06	-1.4e-03	-4.2e-04	-9.0e-04	5.1e-03	3.5e-03		
Model 8	-6.5e-06	-1.5e-04	8.5e-04	7.3e-04	6.7e-03	4.5e-03		
Model 9	-1.7e-05	-1.0e-03	-8.2e-04	-1.1e-03	4.3e-03	2.8e-03		
Model 10	-1.6e-05	-9.9e-04	-7.8e-04	-1.0e-03	-4.0e-03	2.7e-03		
Model 11	-3.2e-05	-1.7e-03	-4.2e-04	-1.2e-03	4.9e-03	3.9e-03	9.9e-03	8.2e-03
Model 12	2.6e-04	2.4e-04	8.6e-04	7.5e-04	6.7e-03	6.2e-03	1.2e-02	1.2e-02
Model 13	-5.5e-05	-1.1e-03	-8.2e-04	-1.2e-03	4.2e-03	3.6e-03	9.3e-03	7.8e-03
Model 14	-4.8e-05	-8.7e-04	-5.7e-04	-8.4e-04	4.3e-03	3.8e-03	9.7e-03	8.9e-03

Where;

σ_i, σ_o : Inside and outside hoop stresses for the part

Δ_i, Δ_o : Inside and outside radial deflection for the part

Non-Linear Analysis versus Linear Analysis

Due to the analyses result shown in Table 2 and Table 3, a decision can be made on the type of analysis and technique which shall be used for shrink fit problems made of multiple fitted parts.
Table 4 demonstrates the comparison between models analyzed. Results of hoop stresses for groups 3 and 4 (as shown in Table 1) were investigated.

The analogy considered in this paper is based on the results shown in Table 2. First a decision has to be taken between staged and non-staged technique for linear analysis. A similar decision action has to be taken for the non-linear analysis. The third decision that has to be taken is whether non-linear or linear analysis for the non-staged technique has to be considered. This procedure is applicable for any group.

Investigation of Group 3 results for model numbers 8-11 leads us to the following

1- Linear analysis with staged technique yields less hoop stresses than non-staged technique between the fitted parts. This can be seen in Models 8 and 9.
2- Non-linear analysis with staged technique yields less hoop stresses than non-staged technique between the fitted parts. This can be seen in Models 10 and 11.
3- Because linear analysis with staged technique yields unpredictable results, a decision was made to investigate the non-linear versus linear analysis with non-staged technique. This comparison yields less hoop stresses for non-linear than linear analysis between the fitted parts. This can be seen in Models 8 and 10.

A similar comparison trend is concluded for Group 4 model numbers 12-15. Figure 10 to Figure 12 illustrate ANSYS results for Model 15 only. In Figure 10, first stage shows that there isn't any stress distribution in the gear and the collar. The second stage shows that there isn't stress distribution in the collar. Finally, third stage shows the stress distribution for all parts, the results support the staging technique that was initially proposed. Hoop stresses in the final stage for each part is shown in Figure 12.

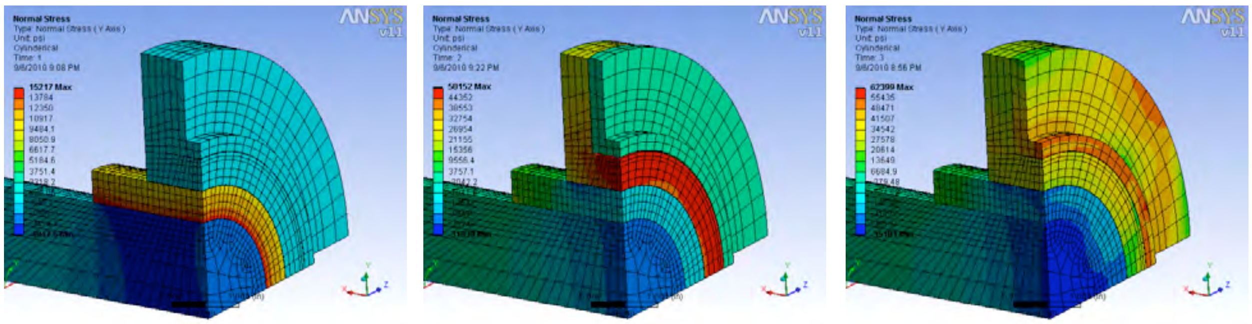
Figure 11 depicts the trend of maximum and minimum stresses and d eflections throughout the proposed stages. For example, the maximum hoop stress proportionally increases between successive stages. Likewise, the minimum hoop stress proportionally decreases between successive stages. The trend for radial deflection is different. Deflection increases in the first two stages. However, once the collar is installed (third stage) the radial deflection decreased by 50% approximately.

These figures represents the final conclusion of the analysis, similar plots can be extracted for all models.

Table 4: Models and Analyses Comparison

Model #	Analysis Classification	No. of Parts	Models Comparison	Shaft		Bushing		Gear		Collar	
				σ_I (ksi)	σ_o (ksi)	σ_I (ksi)	σ_o (ksi)	σ_I (ksi)	σ_o (ksi)	σ_I (ksi)	σ_o (ksi)
Model 8	L-NS	3	8 & 9	L-S	L-NS	L-S	L-S	L-S	L-S		
Model 9	L-S	3									
Model 10	NL-NS	3	10 & 11	NL-S	NL-S	NL-S	NL-S	NL-S	NL-S		
Model 11	NL-S	3									
			8 & 10	NL-NS	NL-NS	L-NS	NL-NS	NL-NS	NL-NS		
Model 12	L-NS	4	12 & 13	L-S	L-S	L-S	L-S	L-S	L-S	L-S	L-S
Model 13	L-S	4									
Model 14	NL-NS	4	14 & 15	NL-S	NL-S	NL-S	NL-S	NL-NS	NL-S	NL-NS	NL-S
Model 15	NL-S	4									
			12 & 14	NL-NS	NL-NS	NL-NS	NL-NS	NL-NS	L-NS	NL-NS	NL-NS

Where;
L-NS: Linear and Non-Staged
L-S: Linear and Staged
NL-NS: Non-Linear and Non-Staged
NL-S: Non-Linear and Staged



Stage 1 Stage 2 Stage 3

Figure 10: Hoop stress distribution for Model 15 assembly (Non-Linear Staged Analysis)

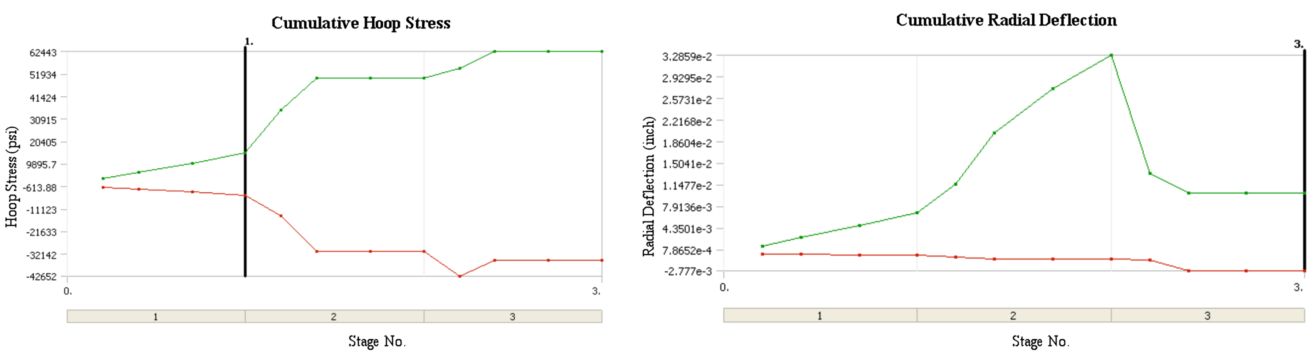


Figure 11: Cumulative Hoop stress and radial deflection for Model 15 throughout all stages

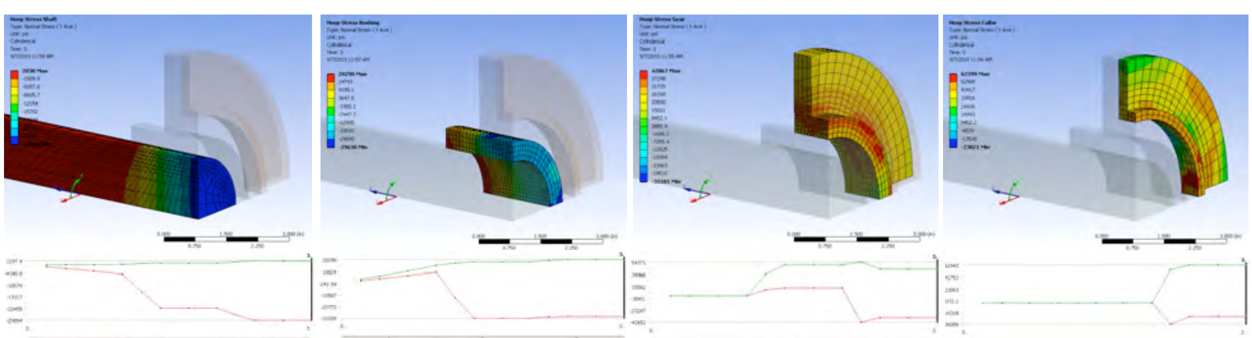


Figure 12: Last stage hoop stress distribution for the parts in Model 15 (Non-Linear Staged Analysis),

Conclusion

Analysis was performed to simulate the shrink fit of many parts using staged technique. ANSYS ensures that considering nonlinear analysis using staged technique gives lower stress and deflection values than using linear model using non staged technique.

References

1- Y.M. Al-Smadi (2008) “Computer Aided Design/Engineering for Trunnion Girder Design on Water Street Bridge”, Proceedings of Heavy Movable Structures Symposium, Orlando, FL.

2- ANSYS, INC. Documentation Manual, Release 11