www.aersys.com	**AERSYS KNOWLEDGE UNIT**	AERSYS-1001

Author:	Asier Ruiz de Aguirre Malaxetxebarria	Date:	31/12/2010

FEM	X	HAND		LIN	X	NOLIN	X	BUCK		FAT		STATIC	X	COMP		MET	X

STRESS CONCENTRATIONS IN METALLIC PARTS

1. INTRODUCTION

When performing a linear analysis of a metallic part that is prone to have stress concentrations (such as those created by holes) the stress peak obtained with the FEM is unrealistic. The plastic behavior of the metallic material will relieve the stress level in the surroundings of the concentration source. The problem is to determine if this stress peak that has been calculated by FEM (or by hand-calculations, i.e. Peterson's handbook) will provide a Reserve Factor above one when taking into account the plasticity. In general remember that this plastic relief will not be applicable to fatigue problems which commonly are dealing with stresses in the elastic range.

The first approach to the problem would be to run the FEM with a non-linear solution (NASTRAN SOL 106) adding the plastic behavior of the material. This solution is reliable and will provide the real stress level in the nearby of the stress concentration and additionally will take into account the effect on the load path due to the stiffness reduction in the area affected by the yielding of the material.

The second solution (based on Neuber hyperbola) is an approximate approach to be used when the level of accuracy desired is not high. The non-linear analysis is not necessary on this approach.

The third possibility is to calculate the stress concentration factor on the plastic range. Usually the stress concentration factor on the elastic range can be easily evaluated by FEM or by using the Peterson's handbook. This Kt can be extrapolated to the plastic range by means of a simple approach.

2. FEM NON-LINEAR ANALYSIS

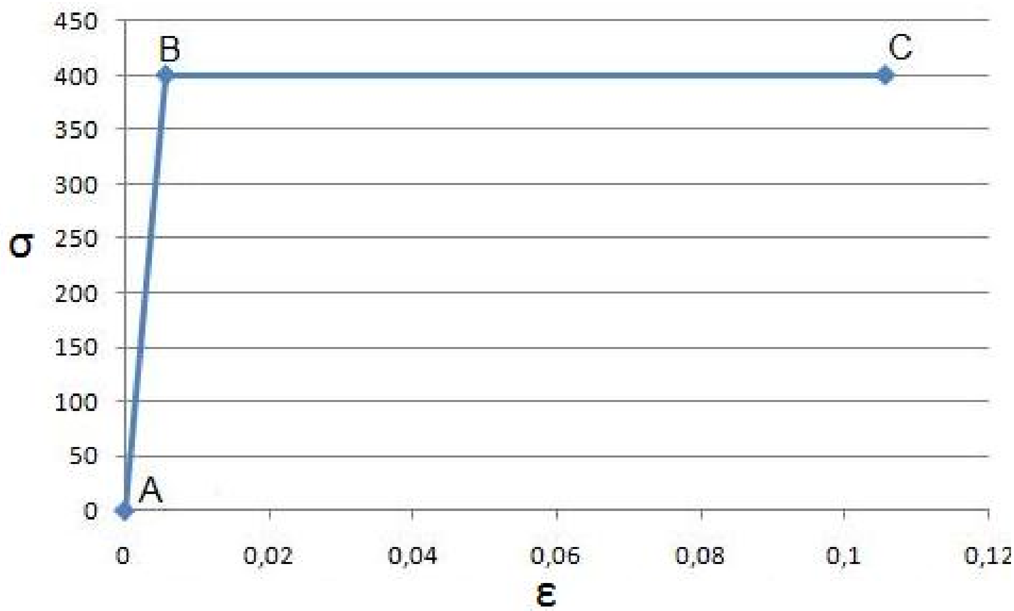
In order to run a non-linear analysis including plasticity on the FEM five changes have to be made on the input of NASTRAN:

- SOL 106 has to be used.
- NLPARM=1 has to be added on the case control (where subcases are defined).
- NLPARM 1 has to be added on the bulk data.
- MATS1 card has to be added. The ID of this card has to be the same that the one on the MAT1 referencing the metallic material with plastic behavior. The third field of this card will reference the ID of a TABLES1 card. The fourth field of this card has to be "NLELAST" ("PLASTIC" option would be used if the analysis would have to take into account the hardening effect of successive loading and

unloading the structure which is not the usual case).

- TABLES1 card has to be added to define the stress-strain curve of the material. The stress-strain curve of the material can be defined by several approaches:

1. Elastic-perfectly plastic behavior:
   This model of the material can be represented by using three points:



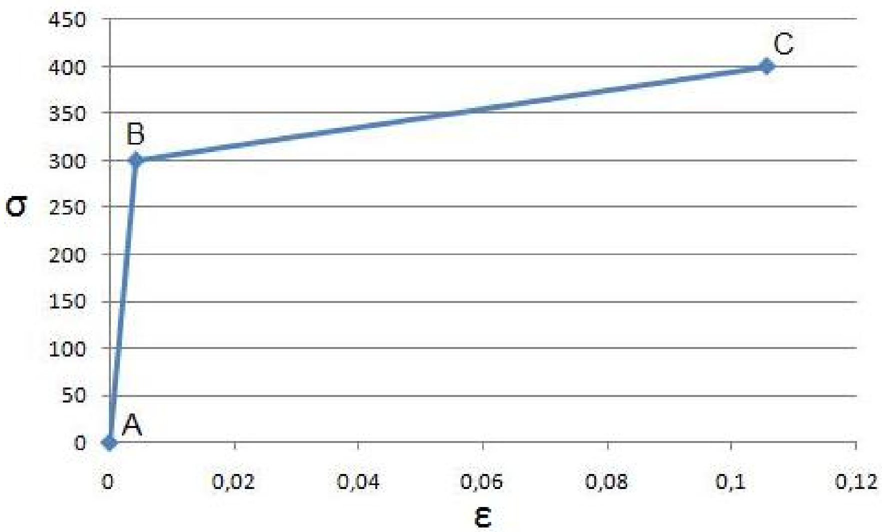
Point	Stress	Strain
A	0	0
B	Ftu	Ftu/E
C	Ftu	Ftu/E+e

Where:
Ftu: Ultimate Strength of the material.
E: Elastic modulus of the material.
e: Ultimate strain of the material.

The main drawback of this model of the material is that the stress cannot be used as a method to evaluate the Reserve Factor (RF) because all the plastic range of the chart has the same stress level. In order to evaluate the criticality of the structure the strain should be used:

$$RF_\varepsilon = \frac{\frac{F_{tu}}{E} + e}{\varepsilon_{FEM}}$$

2. Elastic-Plastic behavior with linear hardening:
   This model of the material can be represented by using three points:



Point	Stress	Strain
A	0	0
B	Fty	Fty/E
C	Ftu	Ftu/E+e

Where:
Fty: Yielding stress of the material.
Ftu: Ultimate Strength of the material
E: Elastic modulus of the material
e: Ultimate strain of the material

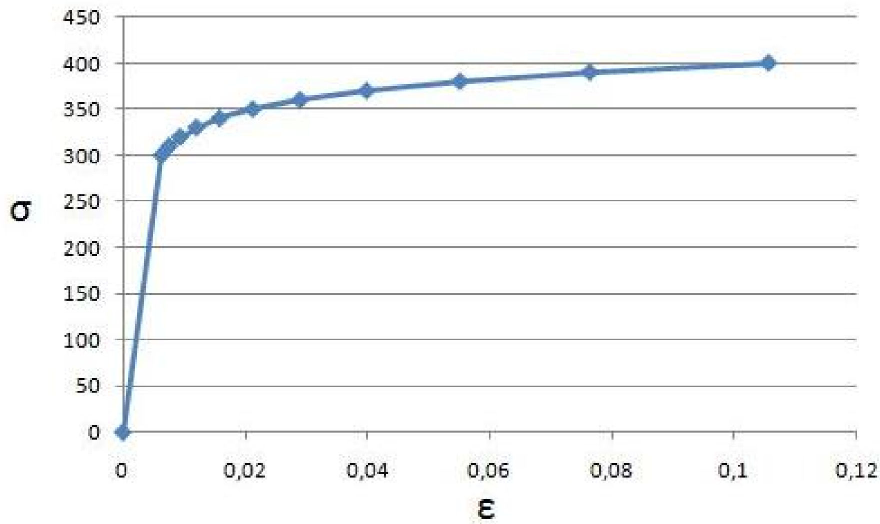
On this model of the material the stress obtained through the FEM can be directly used to evaluate the RF. This RF (as in the previous model) will not be directly related with the loads (as in the usual definition of RF)

$$RF_\sigma = \frac{F_{tu}}{\sigma_{FEM}}$$

The stress to be read in the FEM is the equivalent stress (which is a kind of

Von Mises that NASTRAN uses to search on the material curve) or the stress tensor Von Mises stress. Both can be considered as valid approaches.

3. Ramberg-Osgood Curve:
This model of the material needs several points since the equation of the curve is not poly-lineal. According to the level of accuracy desired the number of points will vary. The best option is to select the number and location of points that will fit correctly the curve of the material by visual inspection.



The equation of the Ramberg-Osgood approach follows:

$$\varepsilon = \frac{\sigma}{E} + 0.002\left(\frac{\sigma}{F_{ty}}\right)^{n}$$

Where:

Fty: Yielding stress of the material.
n: Ramberg-Osgood parameter. This value can be calculated as:

$$n = \frac{ln\left(\frac{e}{0.002}\right)}{ln\left(\frac{F_{tu}}{F_{ty}}\right)}$$

Where:
e: Ultimate strain of the material.
Ftu: Ultimate strength of the material.

On this model of the material the stress obtained through the FEM can be directly used to evaluate the RF. This RF (as in the previous model) will not be directly related with the loads (as in the usual definition of RF)

$$RF_{\sigma} = \frac{F_{tu}}{\sigma_{FEM}}$$

The stress to be read in the FEM is the equivalent stress (which is a kind of Von Mises that NASTRAN uses to search on the material curve) or the stress tensor Von Mises stress. Both can be considered as valid approaches.

The typical NASTRAN input will look like this:

```
SOL 106
CEND
…
NLPARM=1
…
SUBCASE 1
…
…

BEGIN BULK
$111111112222222233333333444444445555555566666666777777777
NLPARM    1
MAT1      1     70000.          0.3
MATS1     1       1     NLELAST
TABLES1   1
          0.0     0.0   0.004286  300.0 0.105714  400.0  ENDT
…
…
```

3. NEUBER APPROACH

The Neuber approach states that the product of the stress and strain in the plastic range will equal to the product of the stress and strain in the elastic solution calculated with a linear approach (since the strain energy should be similar). This way the following relation can be written:

$$\sigma_{NONLINEAR}\varepsilon_{NONLINEAR} = \sigma_{LINEAR}\varepsilon_{LINEAR}$$

Additionally the following expression is applicable to the linear calculations:

$$\varepsilon_{LINEAR} = \frac{\sigma_{LINEAR}}{E}$$

Therefore:

$$\varepsilon_{NONLINEAR} = \frac{(\sigma_{LINEAR})^2}{E\sigma_{NONLINEAR}}$$

Replacing this equation on the Ramberg-Osgood model of the material:

$$\frac{(\sigma_{LINEAR})^2}{E\sigma_{NONLINEAR}} = \frac{\sigma_{NONLINEAR}}{E} + 0.002\left(\frac{\sigma_{NONLINEAR}}{F_{ty}}\right)^n$$

The nonlinear stress can be calculated using the expression above from the constants of the material and the Von Mises stress resulting from a linear analysis (such as linear FEM). The best way to solve this expression is to sweep all the stresses from Fty to Ftu with successive slight increments of stress till the equation is fulfilled. The iterative processes in this kind of equations tend to oscillate and are not recommended.

Once the nonlinear stress is obtained, the Reserve factor can be estimated as:

$$RF_\sigma = \frac{F_{tu}}{\sigma_{NONLINEAR}}$$

Another possibility is to calculate the maximum stress that can be obtained on the linear model with a RF above one. On the limit:

$$\sigma_{LINEAR}^{MAX}\varepsilon_{LINEAR} = F_{tu}\left(\frac{F_{tu}}{E} + e\right)$$

$$\sigma_{LINEAR}^{MAX} = \sqrt{E\left[F_{tu}\left(\frac{F_{tu}}{E} + e\right)\right]}$$

And therefore the RF is calculated as:

$$RF = \frac{\sigma_{LINEAR}^{MAX}}{\sigma_{LINEAR}}$$

This RF is related with the loads since the analysis is linear. The value of the linear stress must be the Von Mises stress to take into account any possible multiaxial stress state.

The application of the Neuber approach is possible if the following assumptions are fulfilled by the structure and loads:

- The nonlinear strain should never be higher than a 10% of the ultimate strain. If this value is passed the accuracy of the method tends to decay quickly:

$$\varepsilon_{NONLINEAR} = \frac{(\sigma_{LINEAR})^2}{E\sigma_{NONLINEAR}} < 0.1e$$

- The area of the cross-section affected by yielding should be lower than a 10% of the total cross-section
- The geometry and the loads allow a redistribution of the load path when the part is on the plastic range.

4. PLASTIC STRESS CONCENTRATION FACTOR

The stress concentration factor in the plastic range can be obtained by using the following general expression:

$$Kt_{plastic} = 1 + (Kt_{elastic} - 1)\frac{E_s}{E_\infty}$$

Where:

$Kt_{plastic}$: Stress concentration factor on the plastic range.
$Kt_{elastic}$: stress concentration factor on the elastic range (calculated or obtained through tables).
E_s: Secant Modulus of the material at the stress level where the Kt has to be evaluated.
$E\infty$: Secant Modulus far from the stress concentration source (in general this value equals to E)

The solution of this expression is not explicit because the value of E_s cannot be evaluated without the value of $Kt_{plastic}$. The expression to evaluate the E_s is:

$$E_s = \frac{E}{1 + \frac{0.002E}{F_{ty}}\left(\frac{\sigma}{F_{ty}}\right)^{n-1}}$$

The new stress will be evaluated as:

$$\sigma = Kt_{plastic}\sigma_\infty$$

Where:

$\sigma\infty$: Stress far from the stress concentration source (Von Mises stress).

The solution of the expressions above (as the Neuber approach) tends to oscillate if iterative methods are used. Hence, it is recommended to sweep all the range of stresses from Fty to Ftu with successive slight increments on the stress till all the equations are fulfilled.

The RF for this approach will be evaluated as:

$$RF_{\sigma} = \frac{F_{tu}}{\sigma}$$

This RF will not be directly related with the loads (as in the usual definition of RF).

The validity of this expression is in general constrained by the same assumptions that the Neuber approach.

- The peak strain should never be higher than a 10% of the ultimate strain. If this value is passed the accuracy of the method tends to decay quickly:

$$\varepsilon = \frac{\sigma}{E} + 0.002\left(\frac{\sigma}{F_{ty}}\right)^{n} < 0.1e$$

- The area of the cross-section affected by yielding should be lower than a 10% of the total cross-section
- The geometry and the loads allow a redistribution of the load path when the part is on the plastic range.