3.2 PIPING BENCHMARK PROBLEMS

COMPUTER ANALYSIS WITH THE CEASEMT FINITE ELEMENT SYSTEM

by
H. Bung, G. Clement, A. Hoffmann
and H. Jakubowicz

This section presents results for the analyses of all three International Piping Benchmark Problems. An inelastic analysis of each problem was performed using a full three-dimensional shell analysis (TRICO code) and a simplified piping analysis based on beam theory (TEDEL code).
PIPING BENCHMARK PROBLEMS

COMPUTER ANALYSIS WITH THE CEASEMT FINITE ELEMENT SYSTEM

H. BUNG, G. CLEMENT, A. HOFFMANN, H. JAKUBOWICZ

Les informations contenues dans ce document sont réservées aux destinataires nommément désignés. Elles ne peuvent recevoir aucune autre diffusion sans l'autorisation expresse du Département des Études Mécaniques et Thermiques.
FOREWORD

In accordance with recommendations, at the 1976 IWGFR specialists' Meeting on High-Temperature Structural Design, three experimental benchmark piping problems were selected. The D.E.M.T (Departement des Etudes Mecaniques et Thermiques) of the French Atomic Energy Commission has contributed to this benchmark effort by computing the three proposed problems.

Two different types of analyses were conducted for each problem. Firstly a full three dimensional shell analysis was performed then a simplified piping analysis method is used. The comparison of these two analyses with the experimental results and between themselves permits not only to assess non linear finite element analysis for piping but also the advantages and limitations of simplified methods.

This report is divided in three parts. The first part presents the three dimensional analyses for the benchmark problems. The second part presents the simplified piping analysis of the same problems. The third part is devoted to the comparison between the two analyses.

The first and second parts present the respectively used computer codes and a brief summary of the theories used in them. Special mention is made of the global method and of the \( e^{*} \) Hoffmann method for non linear analysis. The elbow element of the simplified method is described. Input and output of both codes are presented. Then the three benchmark problems are described with all relevant data for the computation. The finite element idealisation is given and the results are shown on tables and diagrams. The computed results are compared to the measured results.

The third part presents the hypotheses made to explain the observed differences between the three dimensional computation and the measured data. Then are explained the differences
between the two types of analyses. All these explanations are supported by further computations with different shapes, material data or thicknesses. A cost effectiveness analysis is made for the two types of analysis methods.
CONTENTS

FOREWORD. 0.1

PART ONE - PIPING BENCHMARK PROBLEMS ANALYSIS WITH THE THREE DIMENSIONAL SHELL ANALYSIS COMPUTER CODE TRICO.

1.- A three dimensional elastic-plastic-creep shell analysis program.
   1.1.- Introduction. 1.1

   1.2.- Description of the code TRICO.
      1.2.1.- General features. 1.1
      1.2.2.- The global method. 1.2
      1.2.3.- The $\varepsilon^*$-Hoffmann method 1.5
      1.2.4.- Material data input. 1.15
      1.2.5.- Input and output of TRICO. 1.17

   1.3.- Elbow-pipe assembly subjected to in-plane moment loading at 593° C.
      1.3.1.- Problem description. 1.17
      1.3.2.- The finite element idealisation. 1.20
      1.3.3.- Results. 1.22
      1.3.4.- Conclusions. 1.25

   1.4.- Elevated temperature elastic-plastic-creep test of an elbow subjected to in plane moment loading.
      1.4.1.- Problem description. 1.51
      1.4.2.- The finite element idealisation 1.53
      1.4.3.- Results. 1.55
      1.4.4.- Conclusions. 1.55

   1.5.- Room-temperature elastic-plastic response of a thin walled elbow subjected to in plane bending loads.
      1.5.1.- Problem description. 1.69
      1.5.2.- The finite element idealisation 1.70
      1.5.3.- Description of the implementation of large displacements in the $\varepsilon^*$-Hoffmann method for non linear structural analysis 1.73
      1.5.4.- Results 1.76
PART TWO - PIPING BENCHMARK PROBLEMS ANALYSIS WITH A SIMPLIFIED PIPING ANALYSIS METHOD.

2.- A simplified method for non linear piping analysis.
   2.1.- Introduction. 2.1

2.2.- Description of the simplified piping analysis computer code TEDEL. 2.1
   2.2.1.- General features. 2.1
   2.2.2.- The global method applied to pipes. 2.2
   2.2.3.- The elbow element. 2.5
   2.2.4.- Other features. 2.8
   2.2.5.- Input and output of the TEDEL computer code. 2.8

2.3.- Elbow-pipe assembly subjected to in-plane moment loading at 593° C. 2.11
   2.3.1.- The finite element idealisation. 2.11
   2.3.2.- Results. 2.13
   2.3.3.- Conclusions. 2.17

2.4.- Elevated temperature elastic-plastic-creep test of an elbow subjected to in-plane moment loading. 2.43
   2.4.1.- The finite element idealisation. 2.43
   2.4.2.- Results. 2.45

2.5.- Room-temperature elastic-plastic response of a thin walled elbow subjected to in plane bending loads. 2.52
   2.5.1.- The finite element idealisation. 2.52
   2.5.2.- Results. 2.55
   2.5.3.- Conclusions. 2.56

PART THREE - ANALYSIS OF BENCHMARK TESTS AND COMPUTER RESULTS.

3.- Evaluation of the benchmark problems. 3.1
   3.1.- Introduction. 3.1

   3.2.- Influence of material data. 3.2
PART ONE

PIPING BENCHMARK PROBLEMS ANALYSIS WITH THE
THREE DIMENSIONAL SHELL ANALYSIS
COMPUTER CODE TRICO

1 A THREE DIMENSIONAL ELASTIC-PLASTIC-CREEP SHELL ANALYSIS
PROGRAM

1.1 Introduction

The TRICO shell analysis computer code is part of the CEASEMT finite element system. It has access to all of this system capabilities. This system is characterised by a modular architecture especially designed for non linear and dynamic structural problems. A complete library of elements is available as well as a huge capability of pre and post-processors for two and three dimensional problems.

The COCO pre-processor (reference 3) is a mesh generator with its own geometrically oriented language. The TEMPS and ESPACE post-processors (reference 4) are used to display graphically all relevant results respectively in time for a given point and in space for a given time.

1.2 Description of the code TRICO (references 1, 2)

1.2.1 General features

As most of the CEASEMT system programs the TRICO program is based on the finite element method. The TRICO program uses a flat three nodes triangulare constant stress element. Each node has six degrees of freedom. A truss element is also included and can be used in connection with the plate element.
The TRICO program can solve static or dynamic problems. It can search for eigenvalues or give the response to any excitation or to a varying load by time step integration. It can solve Elastic-plastic-creep analysis with large displacements as well as problems in thermoplasticity.

Due to its dynamic storage allocation capability, the TRICO program can solve problems with a large number of unknowns. Its limits being those of the out of core storage of the computer center.

The loading conditions and boundary conditions are numerous and general enough for all engineering needs. It is possible to apply concentrated or distributed forces and moments as well as pressure, dead weight or thermal loads. The displacements can be imposed in any direction. Relations between displacements or between forces and displacements are easily taken in account as well as the symmetry conditions.

1.2.2 The global method (Ref.9)

The basis of this formalism is the notion of generalised stresses and strains introduced by Prager (reference 18). These generalised stresses and strains are placed in duality by a bilinear form expressing the virtual work. The equations of plasticity are then expressed directly in terms of these generalised stresses and strains.

For the plate (or shell) element the generalised stresses are the membrane tensions \( N_1, N_2, N_3 \) and the bending moments \( M_1, M_2, M_3 \). Corresponding to these stresses are the generalised strains defined by membrane strains \( e_1, e_2, e_3 \) and curvature variations \( (\chi_1, \chi_2, \chi_3) \).

To define a generalised yield surface the second order invariants of the generalised stresses deviator are used in conjunction with the Von Mises criterion. The three invariants are
\[ N^2 = N_1^2 + N_2^2 - N_1 N_2 + 3N_3^2 \]
\[ M^2 = M_1^2 + M_2^2 - M_1 M_2 + 3M_3^2 \]
\[ MN \cos \theta = N_1 M_1 + N_2 M_2 - 0.5(M_1 N_2 + M_2 N_1) + 3M_3 N_3 \]

and the yield surface is an expression of the form

\[ F(M, N, \cos \theta, \mu_1, \mu_2, \ldots) = 0 \]

where \( \mu_1, \mu_2, \ldots \) are material history parameters.

Hill's principle for plastic flow applied to these relations produces the following relations

| \[ de_1 \] | \[ de_2 \] | \[ de_3 \] |
| \[ \frac{de_1}{d\lambda} = \frac{3F}{3N} \frac{N_1 - 0.5N_2}{N} + \frac{1}{N} \frac{M_1 - 0.5M_2}{M} \] | \[ \frac{de_2}{d\lambda} = \frac{3F}{3N} \frac{N_2 - 0.5N_1}{N} + \frac{1}{N} \frac{M_2 - 0.5M_1}{M} \] | \[ \frac{de_3}{d\lambda} = \frac{3F}{3N} \frac{N_3}{N} + \frac{1}{N} \frac{3M_3}{M} \] |

| \[ \frac{dx_1}{d\lambda} = \frac{3F}{3M} \frac{M_1 - 0.5M_2}{M} + \frac{1}{M} \frac{N_1 - 0.5N_2}{N} \] | \[ \frac{dx_2}{d\lambda} = \frac{3F}{3M} \frac{M_2 - 0.5M_1}{M} + \frac{1}{M} \frac{N_2 - 0.5N_1}{N} \] | \[ \frac{dx_3}{d\lambda} = \frac{3F}{3M} \frac{3M_3}{M} + \frac{1}{M} \frac{3N_3}{N} \] |

As usually \( d\lambda \) is obtained by identifying the relation between equivalent generalised stress and strain intensities to the tensile curve.
In practice in the computer program these generalised stresses and strains are normalised to stress and strain quantities by taking in account the thickness of the plate or the shell.

The hardening law can be either one of the following rules: isotropic hardening, kinematic hardening or a multi-layer model.

Then the usual initial stress method, with some improvements described below, is applied to solve by increments and iterations these non linear equations. For a load increment $\Delta F$ the static equilibrium equation must obviously be satisfied. This is accomplished by the so called "external iterations". If the associated stress distribution does violate the yield criteria in some part of the structure, the stress must be brought back on the stress-strain curve by allowing plastic flow in that part. This is accomplished by the so called "internal iterations".

To be more specific let $\Delta F$ be the load increment and $\Delta U$ the displacement increment. Then $[K] \{\Delta U\} = \{\Delta F\}$ where $[K]$ is the stiffness matrix. If the yield criterion is violated the internal iterations will produce a plastic flow increment $\Delta \varepsilon_P$. To satisfy equilibrium a load increment

$$\{\Delta F\}_2 = \{\Delta R\} + \{\Delta \varepsilon_P\}_1$$

is computed where $\{\varepsilon_m\} = [B_m] \{U\}$, $\{\varepsilon_m\} = [D_m] \{\varepsilon_m\}$, $m$ is an element number and $V_m$, the volume of the element. And then an external iteration is performed that is to say the

$$[K] \{\Delta U\}_2 = \{\Delta R\} + \{\Delta \varepsilon_P\}_1$$

equation is solved.
The yield criterion is then verified again and so on. The iterations are stopped when \( (\Delta P^n) = (\Delta P^{n-1}) \) or if they differ only by a small amount.

### 1.2.3 The \( \varepsilon^* \)-Hoffmann method

At an equilibrium position the equation

\[
[B]^T \{a_Q\} = \{F_Q\} \quad (1)
\]

is verified. Let \( \{dF\} \) be an infinitesimal increment of load then

\[
[B]^T \{a\} = \{F_Q\} + \{dF\} \quad (2)
\]

if \( \{a\} = \{a_Q\} + \{da\} \) then it can be deduced that

\[
[B]^T \{da\} = \{dF\} \quad (3)
\]

But

\[
\{da\} = [D] \{de_e\} = [D] \{(de_t) - (de_P)\} \quad (4)
\]

and

\[
\{de_t\} = [B] \{du\} \quad (5)
\]

where \( \varepsilon^e, \varepsilon^t, \varepsilon^P \) are the elastic, total and plastic strains and \( u \) the displacement. Finally the incremental equilibrium equation can be written as

\[
\]

The Prandtl-Reuss equation for the Von Mises yield criteria can be written as

\[
\{de_P\} = [M] \cdot \frac{\{\sigma\}}{\varepsilon^*} \cdot \varepsilon^x \quad (7)
\]

where \([M]\) is a constant matrix
thus equations (4) and (7) produces

$$\{ \sigma \} = [D] \{ \varepsilon^r \} - [D] [M] \frac{\partial f}{\partial \varepsilon^*} \varepsilon^*$$

and dividing by $\varepsilon^*$ a fundamental equation is obtained

$$\frac{\partial \sigma}{\partial \varepsilon^*} = \frac{\partial \sigma}{\partial \varepsilon^*} - [A] \frac{\partial f}{\partial \varepsilon^*}$$

with $[A] = [D] [M]$ and $\{ \sigma^* \} = [D] \{ \varepsilon^t \}.$

This differential equation is valid only if $\varepsilon^* \neq 0.$

To summarise the previous discussion the following two equations must be integrated in order to solve the plasticity problem:

\[
\begin{cases}
\text{if } \sigma^*(\sigma + \{d\sigma\}) \leq f(\varepsilon^*) \text{ then } \varepsilon^* = 0 \text{ and } \{d\sigma\} = \{d\varepsilon^r\} = [A] \{d\varepsilon\} \\
\text{if } \sigma^*(\sigma + \{d\sigma\}) > f(\varepsilon^*) \text{ then } \varepsilon^* > 0 \text{ and } \frac{\partial \sigma}{\partial \varepsilon^*} = \frac{\partial \sigma}{\partial \varepsilon^*} - [A] \frac{\partial f}{\partial \varepsilon^*}
\end{cases}
\]

where $f$ is the yield surface.

For a finite increment of load $\{DR\}$ the first external iteration produces the first displacement by
If $\sigma^\star ((\sigma_o^0 + (\Delta \sigma^\star))) \leq f(\varepsilon_o^\star)$ then $\Delta \varepsilon^\star = 0$

and $(\Delta \sigma) = (\Delta \sigma^\star)$.

But if $\sigma^\star ((\sigma_o^0 + (\Delta \sigma^\star))) > f(\varepsilon_o^\star)$ then $\Delta \varepsilon^\star > 0$

and $(\Delta \sigma)$ is given by the relation

$$\sigma^\star ((\sigma_o^0 + (\Delta \sigma)) = f(\varepsilon_o^\star + \Delta \varepsilon^\star)$$

which says only that the stress is on the tensile curve at the end of the load step.

The problem now is how to integrate the differential stress equation between $\varepsilon_o^\star$ and $\varepsilon_o^\star + \Delta \varepsilon^\star$ with

- $\varepsilon_o^\star$ equivalent strain at the beginning of the load step
- $\varepsilon_o^\star + \Delta \varepsilon^\star$ equivalent strain at the end of the load step
- $\{\sigma_o^0\}$ stress at the beginning of the load step
- $\{\sigma_o^0\} + (\Delta \sigma)$ stress at the end of the load step

1.2.3.1 **The fundamental hypothesis**

The value of $(\frac{\partial \sigma^\star}{\partial \varepsilon^\star})$ is not known so it will be supposed to be constant on the $[\varepsilon_o^\star, \varepsilon_o^\star + \Delta \varepsilon^\star]$ interval and equal to $(\frac{\Delta \sigma^\star}{\Delta \varepsilon^\star})$. This consists only to assume that the stress variation for the load step is linear in $\Delta \varepsilon^\star$. 
To solve the differential system

\[
\frac{d\sigma}{\Delta t} = \frac{\Delta \sigma}{\Delta t} - [A] \{\sigma\} \tag{12}
\]

a diagonalisation is performed on the \( A \) matrix whose eigenvalues are \( \lambda_1 = 0 \) and \( \lambda_2 = \lambda_3 = \lambda_4 = \lambda_5 = \lambda_6 = \frac{3}{2} \). The corresponding eigenvectors are

\[
V_1 = \begin{bmatrix}
1/\sqrt{3} \\
1/\sqrt{3} \\
1/\sqrt{3} \\
0 \\
0 \\
0
\end{bmatrix}
V_2 = \begin{bmatrix}
1/\sqrt{2} \\
-1/\sqrt{2} \\
0 \\
0 \\
0 \\
0
\end{bmatrix}
V_3 = \begin{bmatrix}
1/\sqrt{6} \\
1/\sqrt{6} \\
-2/\sqrt{6} \\
0 \\
0 \\
0
\end{bmatrix}
V_4 = \begin{bmatrix}
0 \\
0 \\
0 \\
1 \\
0 \\
0
\end{bmatrix}
V_5 = \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
0 \\
1
\end{bmatrix}
V_6 = \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
0 \\
1
\end{bmatrix}
\]

In this base the \( \{\sigma\} \) vector has \( \sigma_1 \) components and

\[
\{\sigma\} = \sum_{i=1}^{6} \sigma_i \{v_i\}
\]

thus the differential system can be written

\[
\frac{d\sigma_i}{\Delta t} = \frac{\Delta \sigma_i}{\Delta t} - \lambda \frac{\sigma_i}{\sigma}
\]

\[
\frac{d\sigma_i}{\Delta t} = \frac{\Delta \sigma_i}{\Delta t} - \lambda \frac{\sigma_i}{\sigma} \tag{13}
\]

these equations are uncoupled and can be solved separately. The equations are analytically integrated and the solutions included in the computer program. Two cases will be developed: perfect plastic material and hardening material.
136

1.2.3.2 Perfect plastic material

The solutions of the differential equations are

\[ \sigma_i(x^*) = C \exp\left(-\frac{\lambda_i x^*}{k}\right) + \frac{k}{\lambda_i} \Delta \sigma_i \]  

(14)

the C constant being given by the relation

\[ \sigma_i(x^*) = \sigma_{i0} \]  

\[ \sigma_{i0} \] is the stress value at the beginning of the load step expressed in the base \( \{v_i\} \) of eigenvectors of \( [K] \). Then

\[ \sigma_i(x) = (\sigma_{i0} - \frac{k}{\lambda_i} \Delta \sigma_i) \exp\left(-\frac{\lambda_i x^*}{k}\right) + \frac{k}{\lambda_i} \Delta \sigma_i \]  

(15)

at the end of the load step (in fact at \( x^* + \Delta x^* \)), where

\[ \{\Delta \sigma^t\} = \sum_{i=1}^{6} \Delta \sigma_i \{v_i\}. \]

The only unknown in the last equation is \( \Delta x^* \). Its value is determined by writing than the stresses \( \sigma_i \) are on the stress-strain curve. That is

\[ \sigma^* \{\sigma_i\} = k \]  

(16)

The Newton-Raphson iterative method is used to solve this non linear equation in \( \Delta x^* \). Only a few iterations (2 or 3) are necessary. This method is shown schematically on figure 1.2.3.

Fig 1.2.3. Newton-Raphson iterations for \( \sigma^* = k \)
1.2.3.3 Hardening material (isotropic case)

In this case $\sigma^*$ is not anymore a constant in equation (13). In the $[\varepsilon_0^*, \varepsilon_0^* + \Delta \varepsilon^*]$ interval $\sigma^*$ does increase and this variation is accounted for approximatively.

A limited expansion to the first order, in $\Delta \varepsilon^*$, of equation (15) and of the tensile curve is made and this leads to

$$\sigma_1 = \sigma_{10} + \Delta \sigma_1 - \frac{\lambda_1}{k} (\sigma_{10} + \frac{\Delta \sigma_1}{2}) \Delta \varepsilon^* + O(\Delta \varepsilon^*)$$ (17)

and

$$\sigma^* = \sigma_{0}^* + h \Delta \varepsilon^*$$ (18)

where $h = \left(\frac{d \sigma^*}{d \varepsilon^*}\right)_{\varepsilon_0^*}$ is the tangent to the tensile curve at $\varepsilon_0^*$.

This set of equations (17 and 18) is solved in $\Delta \varepsilon^*$ and a point $(\sigma_1^*, \Delta \varepsilon_1^*)$ is obtained as shown on figure 1.2.4.

![Fig 1.2.4 - Solution of equations 17 and 18.](image)

The internal iterations are carried on now on equations (15) and (16) as in the perfect plasticity case but with the mean value $\overline{\sigma} = k = \frac{1}{2} (\sigma_0^* + \sigma_1^*)$ for the load step. This choice does verify the flow equations only in average on the load step but the error which is introduced is very low.
At the end of the internal iterations the stresses are given by \( \{\sigma\} + \{\Delta \sigma\} \) and they are on the stress-strain curve. The new value \( \varepsilon^p + \Delta \varepsilon^p \) is known as well as the plastic deformation variation.

\[
\{\Delta \varepsilon^P\} = [D]^{-1} (\{\Delta \sigma^t\} - \{\Delta \sigma\})
\]

The plastic load \( \{\Delta F^P\} \) necessary to reequilibrate the structure is then given by

\[
\{\Delta F^P\} = \sum_m \int_{V_m} \left[ B_m \right]^T ((\sigma_{m}^t) - (\sigma_{m}^p)) \, dV_m
\]

This force is added to the second member of the equilibrium equation

\[
[K] \{\Delta U\} = \{\Delta R\} + \{\Delta F^P\}
\]

which is solved by external iterations.

1.2.3.4 Convergence criteria. An acceleration method

The external iterations are stopped when \( \{\Delta F^P\} \) does not change any more. That is done in the computer program by the following criteria. Let \( \sigma_n^* \) be the value of \( \sigma^* (\sigma_o^* + \Delta \sigma_n^t - \Delta \sigma_n^P) \) and \( \sigma_{n-1}^* \) the value of \( \sigma^* (\sigma_o^* + \Delta \sigma_{n-1}^t - \Delta \sigma_{n-1}^P) \). Then these iterations are stopped when

\[
P_n = \frac{|\sigma_n^* - \sigma_{n-1}^*|}{\sigma_n^*} \leq p_{\text{max}}
\]

for all the elements and where \( p_{\text{max}} \) is the convergence criteria supplied by the user.

The same method applies to the internal iterations but with \( p_{\text{max}}/2 \) as convergence criteria.
Depending on material characteristics this convergence can be slow. An acceleration method based on the following extrapolation method was then included in the program.

Let \( \Delta \varepsilon^x_n \) be the variation of equivalent plastic deformation obtained at the issue of the \( n^{\text{th}} \) external iteration and the associated internal iterations. The rate of convergence \( r_n \) is defined by

\[
    r_n = \frac{\Delta \varepsilon^x_n \Delta \varepsilon^x_{n-1}}{\Delta \varepsilon^x_{n-1} \Delta \varepsilon^x_{n-2}}.
\]

This rate is supposed to stabilise after only a few iterations. At the user's demand every \( m \) iterations the limit of the \( \Delta \varepsilon^x_n \) sequence is then calculated by

\[
    r = \frac{\Delta \varepsilon^x_m \Delta \varepsilon^x_{m-1}}{\Delta \varepsilon^x_{m-1} \Delta \varepsilon^x_{m-2}}.
\]

and

\[
    \Delta \varepsilon^x = \frac{1}{1-r} (\Delta \varepsilon^x_m - r \Delta \varepsilon^x_{m-1})
\]

which is the limit of the geometric sequence. Then the plastic deformations are modified in the same proportions

\[
    \{(\Delta \varepsilon^p)\} = \frac{\Delta \varepsilon^x}{\Delta \varepsilon^x_m} \{(\Delta \varepsilon^p)\}
\]

(The upper index \( p \) is for plastic) and the corresponding plastic loads \( \{(\Delta F^p)\} \) are used for the following external iterations.

For some problems if this extrapolation method fails every few iterations the tangential stiffness matrix can be calculated at the user's demand.
1.2.3.4 Hardening material (Kinematic hardening)

The yield criteria is written $F(\{\sigma-a\}) = 0$ and the stress-strain curve supposed to be bilinear. The Prager-Ziegler hypothesis is used to determine the hardening parameter $\{q\}$.

$\{\text{do}\} = d\nu (\{\sigma-a\})$

$d\nu$ is obtained by differentiating the yield surface equation and using the plastic flow equations. That is

$$(d\varepsilon^P) = d\varepsilon^P \frac{\partial f}{\partial \{\sigma-a\}}$$

$$\sigma^* = f(\{\sigma-a\})$$

$$d f(\{\sigma-a\}) = 0 \Rightarrow \left( \frac{df}{d(\sigma-a)} \right)^T \{\sigma-a\} = 0$$

and

$$\left( \frac{df}{d(\sigma-a)} \right)^T \{d\varepsilon\} = \left( \frac{df}{d(\sigma-a)} \right)^T \{d\varepsilon\} = \left( \frac{df}{d(\sigma-a)} \right)^T \{\sigma-a\} \ d\nu$$

Then

$$d\nu = \left( \frac{df}{d(\sigma-a)} \right)^T \{d\varepsilon\}$$

But for constant $a$

$$\left( \frac{df}{d(\sigma-a)} \right)^T \{d\varepsilon\} = d\varepsilon^*$$

and

$$d\nu = \frac{d\varepsilon^*}{\sigma^*} = \frac{h \ v}{\sigma^*}$$

where $h$ is the derivative of the tensile curve at $\varepsilon^*$. This modifies the algorithm previously described in the following manner.
\[ \{d\sigma\}_1 = \{d\sigma^t\}_1 - [\mathbf{M}] \frac{d\epsilon^p}{\sigma^*} \{\sigma - \sigma\}_1 \]

\[ \{d\alpha\}_1 = k \frac{d\epsilon^p}{\sigma^*} \{\sigma - \sigma\}_1 \]

\[ \{d(\sigma - \alpha)\}_1 = \{d\sigma^t\}_1 - ([\mathbf{M}] + h [I]) \{\sigma - \alpha\}_1 \frac{d\epsilon^p}{\sigma^*} \]

where \([I]\) is the unit matrix. It is seen that this equation is the equation (8) by changing \(\{\sigma - \alpha\}\) to \(\{\sigma\}\). The same integration method can then be applied. This method gives simultaneously the solutions of the equation of plasticity and the displacements of the yield surface defined by \(\{d\alpha\}\).

1.2.3.5 Application to creep problems

As usually the first step is to write the equation of equilibrium

\[ [\mathbf{B}] \{\sigma\} = \{\mathbf{F}\} \quad (1) \]

and the equation for the stresses

\[ \{\sigma\} = \{\sigma_0\} + [\mathbf{D}] \{\Delta \epsilon^t\} - \{\Delta \epsilon^F\} \quad (2) \]

The creep flow is given by the following equation

\[ \{\Delta \epsilon^F\} = \int_{t_0}^{t_0 + \Delta t} \frac{\Delta \epsilon}{\sigma(t)} \frac{\Delta \epsilon^p}{\sigma^*} dt \quad (3) \]

on the time interval \([t_0, t_0 + \Delta t]\).

Equations (1) and (2) are combined and as above it is obtained

\[ [\mathbf{K}] [\Delta \mathbf{U}]_n = \{\Delta \mathbf{F}\} + [\mathbf{B}] [\mathbf{D}] \{\Delta \epsilon^p\}_0 + [\mathbf{B}] [\mathbf{D}] \{\Delta \epsilon^P\}_{n-1} \quad (4) \]
and the same iterative method is applied. The question here is how do one obtain \( \{ \Delta \varepsilon \} \) from (3). The integration is numerically conducted with the simple equation

\[
\{ \Delta \varepsilon^P \} = \frac{\Delta t}{2} \left[ \left( \frac{3}{2} \frac{\partial f}{\partial \sigma} \right) \varepsilon^*_{t=t_0} + \left( \frac{3}{2} \frac{\partial f}{\partial \sigma} \right) \varepsilon^*_{t=t_0+\Delta t} \right]
\]

and \( \{ \Delta \varepsilon^P \} \) is then put in equation (4). This last equation replaces the internal iterations previously defined. After equilibrium is obtained at the \( n \)th iterate it is necessary to verify that the stress at the end of the time step is the stress which was used to determine \( \{ \Delta \varepsilon^P \} \). If it is not the case a new iteration is performed.

1.2.4 Material data input

For plastic or creep analyses the ability to introduce easily material data in various manners is very important. Generally these data are given point by point, sometimes an analytical curve is assumed and they are also cases where it is necessary to smooth and fit the data by numerical means. In the TRICO program the stress-strain curves are supplied by either two ways: point by point with the other data related to the problem or by a user supplied subroutine where any type of constitutive equation can be accounted for. The point by point data are used by TRICO in conjunction with a quadratic interpolation. The creep data are always given by the user supplied subroutine in the form of the function.

\[
\varepsilon^* = \varphi (\varepsilon^*, \sigma^*, t, T, \ldots, \ldots)
\]

where \( \varepsilon^* \) is the strain intensity, \( \sigma^* \) the stress intensity, \( t \) the time, \( T \) the temperature etc. ... Any argument of this function can be omitted and other arguments can be added. If the creep data are given point by point the user supplied subroutine must be provided with an interpolation scheme. This way of dealing with creep leaves the designer free to choose any hardening rule (time hardening, strain hardening, or other rules).
1.2.5 **Input and output of TRICO**

a) **Input**

The input must always begin with some parameters on the number of nodes and elements and other values, to allocate the computer memory necessary for the problem. All the other data are input in free format fields in any order due to the use of code names to specify the different data categories. The input is generally composed by:

- The mesh: coordinates of the points, numbers defining the elements
- The thicknesses of the elements
- Some material properties:
  . Young modulus
  . Poisson's ratio
  . Stress-strain curves (if given point by point)
- The loads
- The load and time increments
- The boundary conditions
- Parameters for non linear analysis: maximal number of iterations, convergence criteria
- Parameters for selection of the printed output.

b) **Output**

The results are listed or stored on computer files for latter retrieval by post-processor codes. They can be visualized on graphic display units, hard copy or plotters.
These post-processor codes can be used either by the batch processing mode or the interactive time sharing mode.

All input data are printed. Then for each time or load steps the following results are printed:

- number of plastic elements, actual convergence error, maximal stress and strain intensities. Actual loads and time increments. Number of iterations.

- At selected elements: the principal strains and curvature variations.

For selected time or load steps the following results are printed

- The displacements and rotations

- The boundary reactions

- The principal stresses and moments

1.3 Elbow-Pipe Assembly Subjected to In-plane Moment Loading at 593°C

1.3.1 Problem description (ref. 6)

1.3.1.1 Geometry

The Elbow-pipe Assembly tested at Battelle-Columbus Laboratories is a 101.6 mm sched-10 90° elbow with a 152.4 mm bend radius welded to two 324 mm lengths of sched - 10 pipes (Fig. 1.3.1). The elbow wall thicknesses and diameter were measured at selected grid points (table 1.3.1 and Fig. 1.3.2).
<table>
<thead>
<tr>
<th>α</th>
<th>Wall thicknesses in mm</th>
<th>Outside diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>φ=0°</td>
<td>90°</td>
</tr>
<tr>
<td>0°</td>
<td>3.02</td>
<td>3.2</td>
</tr>
<tr>
<td>22.5°</td>
<td>2.74</td>
<td>2.84</td>
</tr>
<tr>
<td>45°</td>
<td>2.76</td>
<td>2.54</td>
</tr>
<tr>
<td>67.5°</td>
<td>3.37</td>
<td>2.81</td>
</tr>
<tr>
<td>90°</td>
<td>2.79</td>
<td>3.07</td>
</tr>
</tbody>
</table>

Table 1.3.1 - Dimensions of elbow

The average wall thickness is 3 mm and average outside diameter is 114.53 mm. The pipe legs have a wall thickness of $3.07 \pm 0.05$ mm, an outside diameter of $114.25 \pm 0.25$ mm and a length of 323.85 mm.

A rigid frame shown on fig. 3.1 is used to apply the loads.

1.3.1.2 Loads and boundary conditions

As system of dead weights was used to load the assembly. These loads produce a constant moment in the assembly. On figure 1.3.3 the loads are noted as $F_1$ and $F_2$. The load histogram is shown on figure 1.3.4. A first load of 1382 N is applied on each loading point producing a bending moment of 843 Nm. The specimen is held at this load for 295 hours. The load is then increased to 1827 N producing a bending moment of 1114 Nm. This load is held for another 44 hours. At this time the test is ended. The temperature during all the test is maintained at $593°C \pm 5.6°C$. 

The Elbow pipe assembly is rigidly fastened at the other end.

1.3.1.3 Material

The elbow and pipe legs are made of type 304 stainless steel. The material properties are given in reference 5.

Young modulus 149 652 N/mm²

Poisson's ratio 0.3

The stress-strain curve at 593°C is given by the following formula

$$\varepsilon_p = 4.59 \times 10^{-41} \sigma^{9.3377}$$

where $\sigma$ is in psi and $\varepsilon_p$ the plastic strain is in/in. The tensile curve values are given below in table 1.3.2 and the corresponding graph is shown on fig. 1.3.5.

<table>
<thead>
<tr>
<th>$\sigma$ (ksi)</th>
<th>$\sigma$ (N/mm²)</th>
<th>$\varepsilon$ (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.5</td>
<td>44.8</td>
<td>0.0002996</td>
</tr>
<tr>
<td>8.5</td>
<td>58.7</td>
<td>0.0006</td>
</tr>
<tr>
<td>9</td>
<td>62.1</td>
<td>0.0008</td>
</tr>
<tr>
<td>10</td>
<td>69</td>
<td>0.0015</td>
</tr>
<tr>
<td>10.5</td>
<td>72.4</td>
<td>0.0021</td>
</tr>
<tr>
<td>11</td>
<td>75.8</td>
<td>0.0030</td>
</tr>
<tr>
<td>11.5</td>
<td>79.4</td>
<td>0.0043</td>
</tr>
<tr>
<td>12</td>
<td>82.8</td>
<td>0.0062</td>
</tr>
<tr>
<td>15</td>
<td>89.7</td>
<td>0.0125</td>
</tr>
<tr>
<td>17</td>
<td>117.2</td>
<td>0.1468</td>
</tr>
</tbody>
</table>

Table 1.3.2 - Tensile curve at 593°C
The creep data are given by the reference 5 table 2.

1.3.2 The finite element idealisation

1.3.2.1 Geometry

Half of the elbow pipe assembly due to symmetry conditions) is meshed with 770 triangular elements and 432 nodes. The mesh, the element numbering and nodal numbering are presented on figures 1.3.6, 1.3.7 and 1.3.8.

Each element is given its appropriate thickness. The equithickness lines are represented on figure 1.3.9.

The frame at the free end was not meshed but represented by a rigid flange. This flange is represented in the mesh by the last row of elements (749 to 770) which were given a 45 mm thickness.

1.3.2.2 Loads and boundary conditions

The forces are introduced directly on the mesh on points number 421 and 432. They are given a value taking in account the effect of the beams length. That is

\[ F_1^* \times R = \frac{F_1}{2} \times 304.8 \text{ mm} \]

where \( R \) is the mean radius of the pipe, and, due to the symmetry condition, only half of the load is applied.

\[ F_1^* = \frac{1382 \times 304.8}{2 \times 55.63} = 3786.03 \text{ N} \]

The same value is given to \( F_2^* \).

By the same formula the second load is obtained as 5005.12 N.
The boundary conditions as mentioned previously are symmetry conditions for the XOY plane and of the clamped type conditions for the fixed pipe end. This was imposed by setting to zero value all the degrees of freedom of nodes 1 to 12 and symmetry condition on nodes 13, 25, ..., 421 and 24, 36, ..., 432.

1.3.2.3 Material

The stress-strain curve was entered point by point with the values of table 1.3.2.

The creep data were replaced by a similar matrix but with strain rates. These rates were calculated from table 1.3.3 by taking the ratio between strain differences and time differences for a given stress level. (For example at a stress of 14 Ksi between the 50th and 100th hour the rate is $89 \times 10^{-7}$ in/in/hour). These rates were included in a user supplied subroutine where linear interpolation between stress levels was programmed.

1.3.2.4 Miscellaneous data

The maximum number of iterations is set to 23 and the convergence criteria $\tau_{\text{max}}$ to $10^{-2}$. The total number of time and load increments is 51 (Fig. 1.3.10). The first five steps are used to impose the initial load at constant time (no hold time). Then the following time table is used:

- between 0 and 10 hours 10 steps
- between 10 and 100 hours 10 steps
- between 100 and 295 hours 14 steps

Then the load is incremented in five steps at constant time and between the 295th and 339th five time steps are used. The unloading is made in two load steps at 339 hours.
1.3.3 Results

1.3.3.1 Displacements

Due to the idealisation some additional computation are necessary to provide typical displacements that are comparable to the set presented in the Benchmark problem. The quantities $\delta_1$, $\delta_2$, $\delta_3$, $\delta_y$ and $\theta$ as defined in reference 6 and on figure 1.3.11 are obtained by taking the mean values of $u_A$ and $v_A$, the displacements in the x and y directions respectively, of the points 425 and 426 of the mesh. The following formulas are applied:

\[
\begin{align*}
\delta_1 &= v_A - 304.8 \theta_A \\
\delta_2 &= -v_A - 304.8 \theta_A \\
\delta_3 &= -165.1 \theta_A + u_A \\
\delta_y &= v_A \\
\theta &= -\theta_A
\end{align*}
\]

The results are presented on table 1.3.3 and figures 1.3.12, 1.3.13, 1.3.14, 1.3.15, 1.3.16. The comparison with experimental results shows that these computations over estimate the displacement. This difference is supposed to come for the material data which do not seem to take in account the metallurgical charges of the elbow material which was rolled, welded and hot bended. Some complementary work was conducted with ASME material data. These results are presented in Part 3 of this report.
Tableau 1.3.3 - Calculated displacements

1.3.3.2 Strains

The computed strains are given for the strain gages located on figure 1.3.17. The table 1.3.4 gives this values in micro deformation in function of time. The number of the gage corresponds to the number on figure 1.3.17.

<table>
<thead>
<tr>
<th>Step number</th>
<th>time (hours)</th>
<th>δ₁ (mm)</th>
<th>δ₂ (mm)</th>
<th>δ₃ (mm)</th>
<th>δᵧ (mm)</th>
<th>θ (10⁻³rd)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>5.71</td>
<td>3.526</td>
<td>8.7</td>
<td>1.092</td>
<td>15.15</td>
</tr>
<tr>
<td>10</td>
<td>5</td>
<td>6.251</td>
<td>3.857</td>
<td>9.518</td>
<td>1.197</td>
<td>16.58</td>
</tr>
<tr>
<td>15</td>
<td>10</td>
<td>6.507</td>
<td>4.012</td>
<td>9.903</td>
<td>1.248</td>
<td>17.25</td>
</tr>
<tr>
<td>20</td>
<td>55</td>
<td>7.373</td>
<td>4.55</td>
<td>11.225</td>
<td>1.411</td>
<td>19.56</td>
</tr>
<tr>
<td>25</td>
<td>100</td>
<td>7.732</td>
<td>4.775</td>
<td>11.776</td>
<td>1.478</td>
<td>20.52</td>
</tr>
<tr>
<td>30</td>
<td>169.64</td>
<td>8.157</td>
<td>5.039</td>
<td>12.425</td>
<td>1.559</td>
<td>21.65</td>
</tr>
<tr>
<td>35</td>
<td>293.29</td>
<td>8.491</td>
<td>5.246</td>
<td>12.933</td>
<td>1.623</td>
<td>22.53</td>
</tr>
<tr>
<td>39</td>
<td>295</td>
<td>8.685</td>
<td>5.365</td>
<td>13.226</td>
<td>1.66</td>
<td>23.05</td>
</tr>
<tr>
<td>44</td>
<td>295</td>
<td>14.956</td>
<td>9.465</td>
<td>23.079</td>
<td>2.745</td>
<td>40.06</td>
</tr>
<tr>
<td>46</td>
<td>312.26</td>
<td>15.078</td>
<td>9.54</td>
<td>23.264</td>
<td>2.769</td>
<td>40.38</td>
</tr>
<tr>
<td>49</td>
<td>339</td>
<td>15.262</td>
<td>9.652</td>
<td>23.543</td>
<td>2.805</td>
<td>40.87</td>
</tr>
<tr>
<td>Step number</td>
<td>time (hours)</td>
<td>5C</td>
<td>6C</td>
<td>7L</td>
<td>10C</td>
<td>11L</td>
</tr>
<tr>
<td>------------</td>
<td>--------------</td>
<td>------</td>
<td>------</td>
<td>------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>5</td>
<td>0</td>
<td>-1369</td>
<td>452</td>
<td>194</td>
<td>-1721</td>
<td>-478</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-1412</td>
<td>467</td>
<td>202</td>
<td>-1766</td>
<td>-493</td>
</tr>
<tr>
<td>8</td>
<td>3</td>
<td>-1453</td>
<td>481</td>
<td>210</td>
<td>-1810</td>
<td>-507</td>
</tr>
<tr>
<td>10</td>
<td>5</td>
<td>-1483</td>
<td>491</td>
<td>216</td>
<td>-1843</td>
<td>-518</td>
</tr>
<tr>
<td>15</td>
<td>10</td>
<td>-1536</td>
<td>510</td>
<td>227</td>
<td>-2028</td>
<td>-522</td>
</tr>
<tr>
<td>17</td>
<td>28</td>
<td>-1642</td>
<td>547</td>
<td>247</td>
<td>-2102</td>
<td>-578</td>
</tr>
<tr>
<td>20</td>
<td>55</td>
<td>-1734</td>
<td>577</td>
<td>262</td>
<td>-2142</td>
<td>-615</td>
</tr>
<tr>
<td>25</td>
<td>100</td>
<td>-1820</td>
<td>606</td>
<td>277</td>
<td>-2254</td>
<td>-651</td>
</tr>
<tr>
<td>30</td>
<td>169.6</td>
<td>-1915.5</td>
<td>638</td>
<td>295</td>
<td>-2369.5</td>
<td>-689</td>
</tr>
<tr>
<td>35</td>
<td>239</td>
<td>-1990</td>
<td>664</td>
<td>309</td>
<td>-2462</td>
<td>-719</td>
</tr>
<tr>
<td>39</td>
<td>295</td>
<td>-2033</td>
<td>679</td>
<td>317</td>
<td>-2517</td>
<td>-737</td>
</tr>
<tr>
<td>44</td>
<td>297</td>
<td>-3880</td>
<td>1186</td>
<td>542</td>
<td>-4959</td>
<td>-1520</td>
</tr>
<tr>
<td>46</td>
<td>312.6</td>
<td>-3906</td>
<td>1195</td>
<td>548</td>
<td>-4988</td>
<td>-1530</td>
</tr>
<tr>
<td>49</td>
<td>339</td>
<td>-3946</td>
<td>1208</td>
<td>556</td>
<td>-5033.5</td>
<td>-1546</td>
</tr>
<tr>
<td>51</td>
<td>339</td>
<td>-3030</td>
<td>887.1</td>
<td>387</td>
<td>-4025</td>
<td>-1336</td>
</tr>
</tbody>
</table>

Table 1.3.4 Calculated strains

1.3.3.3 Stresses

The stress distribution is represented by the Von Mises stress intensity isocurves. A map of these isocurves is given at the beginning and at the end of each load. Figure 1.3.18 for the initial load, figure 1.3.19 after 295 h, figure 1.3.20 at 295 h for the second load and figure 1.3.21 after 44 hours of hold time for the second load. The same maps are shown in close up by figures 1.3.22 to 1.3.25.
1.3.4 Conclusions

The results presented for the Elbow-pipe assembly benchmark problem are very different from the measured values as given in reference 6. The deflection curves and strain curves in function of time have the same shape but the end values differ by a factor of 2 on total displacement (or a factor of 3 on creep displacement). Does the material data used in the computation correspond to the elbow material which was subjected to some metallurgical charges during the elbow manufacturing?
3.2 PIPING BENCHMARK PROBLEMS

COMPUTER ANALYSIS WITH THE CEASEMT FINITE ELEMENT SYSTEM

by

H. Bung, G. Clement, A. Hoffmann
and H. Jakubowicz

This section presents results for the analyses of all three International Piping Benchmark Problems. An inelastic analysis of each problem was performed using a full three-dimensional shell analysis (TRICO code) and a simplified piping analysis based on beam theory (TEDEL code).
PIPING BENCHMARK PROBLEMS

COMPUTER ANALYSIS WITH THE CEASENT FINITE ELEMENT SYSTEM

H. BUNG, G. CLEMENT, A. HOFFMANN, H. JAKUBOWICZ
In accordance with recommendations, at the 1976 IWGFR specialists' Meeting on High-Temperature Structural Design, three experimental benchmark piping problems were selected. The D.E.M.T (Departement des Etudes Mecaniques et Thermiques) of the French Atomic Energy Commission has contributed to this benchmark effort by computing the three proposed problems.

Two different types of analyses were conducted for each problem. Firstly a full three dimensional shell analysis was performed then a simplified piping analysis method is used. The comparison of these two analyses with the experimental results and between themselves permits not only to assess non linear finite element analysis for piping but also the advantages and limitations of simplified methods.

This report is divided in three parts. The first part presents the three dimensional analyses for the benchmark problems. The second part presents the simplified piping analysis of the same problems. The third part is devoted to the comparison between the two analyses.

The first and second parts present the respectively used computer codes and a brief summary of the theories used in them. Special mention is made of the global method and of the $\varepsilon^2$-Hoffmann method for non linear analysis. The elbow element of the simplified method is described. Input and output of both codes are presented. Then the three benchmark problems are described with all relevant data for the computation. The finite element idealisation is given and the results are shown on tables and diagrams. The computed results are compared to the measured results.

The third part presents the hypotheses made to explain the observed differences between the three dimensional computation and the measured data. Then are explained the differences
between the two types of analyses. All these explanations are supported by further computations with different shapes, material data or thicknesses. A cost effectiveness analysis is made for the two types of analysis methods.
CONTENTS

FOREWORD. 0.1

PART ONE - PIPING BENCHMARK PROBLEMS ANALYSIS WITH THE THREE DIMENSIONAL SHELL ANALYSIS COMPUTER CODE TRICO.

1.- A three dimensional elastic-plastic-creep shell analysis program.
   1.1.- Introduction. 1.1
   1.2.- Description of the code TRICO. 1.1
      1.2.1.- General features. 1.1
      1.2.2.- The global method. 1.2
      1.2.3.- The $\varepsilon^*$-Hoffmann method 1.5
      1.2.4.- Material data input. 1.15
      1.2.5.- Input and output of TRICO. 1.17
   1.3.- Elbow-pipe assembly subjected to in-plane moment loading at 593° C.
      1.3.1.- Problem description. 1.17
      1.3.2.- The finite element idealisation. 1.20
      1.3.3.- Results. 1.22
      1.3.4.- Conclusions. 1.25
   1.4.- Elevated temperature elastic-plastic-creep test of an elbow subjected to in plane moment loading.
      1.4.1.- Problem description. 1.51
      1.4.2.- The finite element idealisation 1.53
      1.4.3.- Results. 1.55
      1.4.4.- Conclusions. 1.55
   1.5.- Room-temperature elastic-plastic response of a thin walled elbow subjected to in plane bending loads.
      1.5.1.- Problem description. 1.69
      1.5.2.- The finite element idealisation 1.70
      1.5.3.- Description of the implementation of large displacements in the $\varepsilon^*$-Hoffmann method for non linear structural analysis 1.73
      1.5.4.- Results 1.76
PART TWO - PIPING BENCHMARK PROBLEMS ANALYSIS WITH A SIMPLIFIED PIPING ANALYSIS METHOD.

2.- A simplified method for non linear piping analysis.
   2.1.- Introduction.  
   2.2.- Description of the simplified piping analysis computer code TEDEL.  
      2.2.1.- General features.  
      2.2.2.- The global method applied to pipes.  
      2.2.3.- The elbow element.  
      2.2.4.- Other features.  
      2.2.5.- Input and output of the TEDEL computer code.  

2.3.- Elbow-pipe assembly subjected to in-plane moment loading at 593° C.  
   2.3.1.- The finite element idealisation.  
   2.3.2.- Results.  
   2.3.3.- Conclusions.  

2.4.- Elevated temperature elastic-plastic-creep test of an elbow subjected to in-plane moment loading.  
   2.4.1.- The finite element idealisation.  
   2.4.2.- Results.  

2.5.- Room-temperature elastic-plastic response of a thin walled elbow subjected to in plane bending loads.  
   2.5.1.- The finite element idealisation.  
   2.5.2.- Results.  
   2.5.3.- Conclusions.  

PART THREE - ANALYSIS OF BENCHMARK TESTS AND COMPUTER RESULTS.

3.- Evaluation of the benchmark problems.  
   3.1.- Introduction.  
   3.2.- Influence of material data.
3.3 - Influence of straight parts on elbows. 3.7

3.4 - Influence of geometrical data. 3.11

3.5 - Cost effectiveness. 3.17

3.6 - Closure. 3.19

4 - References. 4.1
PART ONE

PIPING BENCHMARK PROBLEMS ANALYSIS WITH THE

THREE DIMENSIONAL SHELL ANALYSIS

COMPUTER CODE TRICO

1 A THREE DIMENSIONAL ELASTIC-PLASTIC-CREEP SHELL ANALYSIS PROGRAM

1.1 Introduction

The TRICO shell analysis computer code is part of the CEASEMT finite element system. It has access to all of this system capabilities. This system is characterized by a modular architecture especially designed for non-linear and dynamic structural problems. A complete library of elements is available as well as a huge capability of pre and post-processors for two and three dimensional problems.

The COCO pre-processor (reference 3) is a mesh generator with its own geometrically oriented language. The TEMPS and ESPACE post-processors (reference 4) are used to display graphically all relevant results respectively in time for a given point and in space for a given time.

1.2 Description of the code TRICO (references 1, 2)

1.2.1 General features

As most of the CEASEMT system programs the TRICO program is based on the finite element method. The TRICO program uses a flat three nodes triangular constant stress element. Each node has six degrees of freedom. A truss element is also included and can be used in connection with the plate element.
The TRICO program can solve static or dynamic problems. It can search for eigenvalues or give the response to any excitation or to a varying load by time step integration. It can solve Elastic-plastic-creep analysis with large displacements as well as problems in thermoplasticity.

Due to its dynamic storage allocation capability, the TRICO program can solve problems with a large number of unknowns. Its limits being those of the out of core storage of the computer center.

The loading conditions and boundary conditions are numerous and general enough for all engineering needs. It is possible to apply concentrated or distributed forces and moments as well as pressure, dead weight or thermal loads. The displacements can be imposed in any direction. Relations between displacements or between forces and displacements are easily taken in account as well as the symmetry conditions.

1.2.2 The global method (Ref.9)

The basis of this formalism is the notion of generalised stresses and strains introduced by Prager (reference 18). These generalised stresses and strains are placed in duality by a bilinear form expressing the virtual work. The equations of plasticity are then expressed directly in terms of these generalised stresses and strains.

For the plate (or shell) element the generalised stresses are the membrane tensions \( (N_1, N_2, N_3) \) and the bending moments \( (M_1, M_2, M_3) \). Corresponding to these stresses are the generalised strains defined by membrane strains \( (e_1, e_2, e_3) \) and curvature variations \( (\chi_1, \chi_2, \chi_3) \).

To define a generalised yield surface the second order invariants of the generalised stresses deviator are used in conjunction with the Von Mises criterion. The three invariants are
\[
N^2 = N_1^2 + N_2^2 - N_1N_2 + 3N_3^2
\]
\[
M^2 = M_1^2 + M_2^2 - M_1M_2 + 3M_3^2
\]
\[
MN \cos \theta = N_1M_1 + N_2M_2 - 0.5(M_1N_2 + M_2N_1) + 3M_3N_3
\]

and the yield surface is an expression of the form
\[
F(M,N,\cos \theta, \mu_1, \mu_2, \ldots) = 0
\]

where \( \mu_1, \mu_2, \ldots \) are material history parameters.

Hill's principle for plastic flow applied to these relations produces the following relations

\[
\frac{d \epsilon_1}{d \lambda} = \frac{3F}{3N} \frac{N_1 - 0.5N_2}{N} + \frac{3F}{3M} \frac{M_1 - 0.5M_2}{M} \\
\frac{d \epsilon_2}{d \lambda} = \frac{3F}{3N} \frac{N_2 - 0.5N_1}{N} + \frac{3F}{3M} \frac{M_2 - 0.5M_1}{M} \\
\frac{d \epsilon_3}{d \lambda} = \frac{3F}{3N} \frac{3N_3}{N} + \frac{3F}{3M} \frac{3M_3}{M} \\
\frac{d \chi_1}{d \lambda} = \frac{3F}{3M} \frac{M_1 - 0.5M_2}{M} + \frac{3F}{3N} \frac{N_1 - 0.5N_2}{N} \\
\frac{d \chi_2}{d \lambda} = \frac{3F}{3M} \frac{M_2 - 0.5M_1}{M} + \frac{3F}{3N} \frac{N_2 - 0.5N_1}{N} \\
\frac{d \chi_3}{d \lambda} = \frac{3F}{3M} \frac{3M_3}{M} + \frac{3F}{3N} \frac{3N_3}{N}
\]

As usually \( d \lambda \) is obtained by identifying the relation between equivalent generalised stress and strain intensities to the tensile curve.
In practice in the computer program these generalised stresses and strains are normalised to stress and strain quantities by taking into account the thickness of the plate or the shell.

The hardening law can be either one of the following rules: isotropic hardening, kinematic hardening or a multi-layer model.

Then the usual initial stress method, with some improvements described below, is applied to solve by increments and iterations these non-linear equations. For a load increment $\Delta F$ the static equilibrium equation must obviously be satisfied.

This is accomplished by the so called "external iterations". If the associated stress distribution does violate the yield criteria in some part of the structure, the stress must be brought back on the stress-strain curve by allowing plastic flow in that part. This is accomplished by the so called "internal iterations".

To be more specific, let $\Delta F$ be the load increment and $\Delta U$ the displacement increment. Then $[K] \{\Delta U\} = \{\Delta F\}$ where $[K]$ is the stiffness matrix. If the yield criterion is violated the internal iterations will produce a plastic flow increment $\Delta \varepsilon^P$. To satisfy equilibrium a load increment is computed where $\{\varepsilon_m\} = [B_m] \{U\}$; $\{\sigma_m\} = [D_m] \{\varepsilon_m\}$, $m$ is an element number and $V_m$, the volume of the element. And then an external iteration is performed that is to say the

$$[K] \{\Delta U\}_2 = \{\Delta R\} + [\Delta F^P]_1$$

equation is solved.
The yield criterion is then verified again and so on. The iterations are stopped when \( \{ \Delta P \}^n = \{ \Delta P \}^{n-1} \) or if they differ only by a small amount.

1.2.3 The \( \varepsilon^*-Hoffmann \) method

At an equilibrium position the equation

\[
[B]^T \{ \sigma \} = \{ F \}
\]

is verified. Let \( \{ dF \} \) be an infinitesimal increment of load then

\[
[B]^T \{ \sigma \} = \{ F \} + \{ dF \}
\]

if \( \{ \sigma \} = \{ \sigma_0 \} + \{ d\sigma \} \) then it can be deduced that

\[
[B]^T \{ d\sigma \} = \{ dF \}
\]

But

\[
\{ d\sigma \} = [D] \{ de^e \} = [D] (\{ de^t \} - \{ de^p \})
\]

and

\[
\{ de^t \} = [B] \{ du \}
\]

where \( e^e, e^t, e^p \) are the elastic, total and plastic strains and \( u \) the displacement. Finally the incremental equilibrium equation can be written as

\[
\]

The Prandtl-Reuss equation for the Von Mises yield criteria can be written as

\[
\{ de^p \} = [M] \frac{\{ \sigma \}}{\sigma^*} de^e
\]

where \([M]\) is a constant matrix.
thus equations (4) and (7) produces

\[ (\sigma) = [\sigma] \{d\varepsilon_t^*\} - [\sigma] [M] \{\sigma\}^* \varepsilon^* \]  

and dividing by \( \varepsilon^* \) a fundamental equation is obtained

\[ \frac{d\sigma}{d\varepsilon^*} = \{d\sigma_t^*\} - [A] \{\sigma\}^* \]  

with \([A] = [D] [M]\) and \(\{d\sigma_t^*\} = [D] \{d\varepsilon_t^*\}\).

This differential equation is valid only if \( \varepsilon^* \neq 0 \).

To summarise the previous discussion the following two equations must be integrated in order to solve the plasticity problem:

\[
\begin{cases}
\text{if } \sigma^*(\{\sigma\} + \{d\sigma\}) \leq f(\varepsilon^*) \text{ then } \varepsilon^* = 0 \text{ and } \{d\sigma\} = [A] \{du\} \\
\text{if } \sigma^*(\{\sigma\} + \{d\sigma\}) > f(\varepsilon^*) \text{ then } \varepsilon^* > 0 \text{ and } \frac{d\sigma}{d\varepsilon^*} = \{d\sigma_t^*\} - [A] \{\sigma\}^* \varepsilon^*
\end{cases}
\]

where \( f \) is the yield surface.

For a finite increment of load \( [dR] \) the first external iteration produces the first displacement by
and the stress increment is known by

\[
\{\Delta \sigma^t\} = [A] \{\Delta u\}_1
\]

(11)

If \(\sigma^* (\{\sigma_o\} + \{\Delta \sigma^t\}) \leq f(\varepsilon^*_o)\) then \(\Delta \varepsilon^* = 0\)

and \(\{\Delta \sigma\} = \{\Delta \sigma^t\}\).

But if \(\sigma^* (\{\sigma_o\} + \{\Delta \sigma^t\}) > f(\varepsilon^*_o)\) then \(\Delta \varepsilon^* > 0\)

and \(\{\Delta \sigma\}\) is given by the relation

\[
\sigma^* (\{\sigma_o\} + \{\Delta \sigma\}) = f(\varepsilon^*_o + \Delta \varepsilon^*)
\]

which says only that the stress is on the tensile curve at the end of the load step.

The problem now is how to integrate the differential stress equation between \(\varepsilon_o^*\) and \(\varepsilon_o^* + \Delta \varepsilon^*\) with

. \(\varepsilon_o^*\) equivalent strain at the beginning of the load step

. \(\varepsilon_o^* + \Delta \varepsilon^*\) equivalent strain at the end of the load step

. \(\{\sigma_o\}\) stress at the beginning of the load step

. \(\{\sigma_o\} + \{\Delta \sigma\}\) stress at the end of the load step

1.2.3.1 The fundamental hypothesis

The value of \(\frac{\Delta \sigma_t}{\Delta \varepsilon_t}\) is not known so it will be supposed to be constant on the interval and equal to \(\frac{\Delta \sigma_t}{\Delta \varepsilon_t}\). This consists only to assume that the stress variation for the load step is linear in \(\Delta \varepsilon^*\).
To solve the differential system

\[
\frac{d\sigma}{d\varepsilon} = \Delta \sigma - [A] \frac{\sigma}{\sigma^*}
\]  

(12)

a diagonalisation is performed on the \( A \) matrix whose eigenvalues are \( \lambda_1 = 0 \) and \( \lambda_2 = \lambda_3 = \lambda_4 = \lambda_5 = \lambda_6 = \frac{3}{2} \). The corresponding eigenvectors are

\[
\begin{align*}
V_1 &= \begin{bmatrix} 1/\sqrt{3} \\ 1/\sqrt{3} \\ 1/\sqrt{3} \\ 0 \\ 0 \\ 0 \end{bmatrix} \\
V_2 &= \begin{bmatrix} 1/\sqrt{2} \\ 0 \\ -1/\sqrt{2} \\ 0 \\ 0 \\ 0 \end{bmatrix} \\
V_3 &= \begin{bmatrix} 1/\sqrt{6} \\ 1/\sqrt{6} \\ -2/\sqrt{6} \\ 0 \\ 0 \\ 0 \end{bmatrix} \\
V_4 &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 1 \end{bmatrix} \\
V_5 &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 1 \\ 0 \end{bmatrix} \\
V_6 &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 1 \end{bmatrix}
\end{align*}
\]

In this base the \( \{\sigma\} \) vector has \( \sigma_i \) components and

\[
\{\sigma\} = \sum_{i=1}^{6} \sigma_i \{v_i\}
\]

thus the differential system can be written

\[
\frac{d\sigma_i}{d\varepsilon} = \frac{\Delta \sigma_i}{\Delta \varepsilon} - \lambda \frac{\sigma_i}{\sigma^*}
\]  

(13)

these equations are uncoupled and can be solved separately.

The equations are analytically integrated and the solutions included in the computer program. Two cases will be developed: perfect plastic material and hardening material.
1.2.3.2 Perfect plastic material

The solutions of the differential equations are

\[ \sigma_i(\epsilon^*) = C \exp(-\frac{\lambda_i \epsilon^*}{k}) + \frac{k}{\lambda_i} \frac{\Delta \sigma_i}{\Delta \epsilon^*} \]  

The constant \( C \) being given by the relation

\[ \sigma_i(\epsilon_o^*) = \sigma_i(\epsilon_0) \]

\( \sigma_i(\epsilon) \) is the stress value at the beginning of the load step expressed in the base \( \{v_i\} \) of eigenvectors of \( \mathbb{A} \). Then

\[ \sigma_i(\epsilon^*) = (\sigma_i(\epsilon_0) - \frac{k}{\lambda_i} \frac{\Delta \sigma_i}{\Delta \epsilon^*}) \exp(\frac{\lambda_i}{k} (\epsilon_o^* - \epsilon_0^*)) + \frac{\Delta \sigma_i}{\Delta \epsilon^*} \]  

At the end of the load step (in fact at \( \epsilon_o^* + \Delta \epsilon^* \)), where

\[ \{\Delta \sigma^t\} = \sum_{i=1}^{6} \Delta \sigma_{i}^t \{v_i\} \]

The only unknown in the last equation is \( \Delta \epsilon^* \). Its value is determined by writing that the stresses \( \sigma^* \) are on the stress-strain curve. That is

\[ \sigma^* \{\sigma_i\} = k \]

The Newton-Raphson iterative method is used to solve this non-linear equation in \( \Delta \epsilon^* \). Only a few iterations (2 or 3) are necessary. This method is shown schematically on figure 1.2.3.

![Fig 1.2.3. Newton-Raphson iterations for \( \sigma^* = k \)](image)
1.2.3.3 Hardening material (isotropic case)

In this case $\sigma^*$ is not any more a constant in equation (13). In the $[\epsilon^*_0, \epsilon^*_0 + \Delta \epsilon^*]$ interval $\sigma^*$ does increase and this variation is accounted for approximatively.

A limited expansion to the first order, in $\Delta \epsilon^*$, of equation (15) and of the tensile curve is made and this leads to

$$\sigma_1 = \sigma_1^0 + \Delta \sigma_1^* - \frac{\lambda_1}{k} \left( \sigma_1^0 + \frac{\Delta \sigma_1^*}{2} \right) \Delta \epsilon^* + O(\Delta \epsilon^*)$$  \hspace{1cm} (17)

and

$$\sigma^* = \sigma_0^* + h \Delta \epsilon^*$$  \hspace{1cm} (18)

where $h = \left. \frac{d\sigma^*}{d\epsilon^*} \right|_{\epsilon^*_0}$ is the tangent to the tensile curve at $\epsilon^*_0$.

This set of equations (17 and 18) is solved in $\Delta \epsilon^*$ and a point $(\sigma_1^*, \Delta \epsilon_1^*)$ is obtained as shown on figure 1.2.4.

![Fig 1.2.4 - Solution of equations 17 and 18.](image)

The internal iterations are carried on now on equations (15) and (16) as in the perfect plasticity case but with the mean value $\bar{\sigma}^* = k = \frac{1}{2}(\sigma_0^* + \sigma_1^*)$ for the load step. This choice does verify the flow equations only in average on the load step but the error which is introduced is very low.
At the end of the internal iterations the stresses are given by \( \{\sigma\} + \{\Delta \sigma\} \) and they are on the stress-strain curve. The new value \( \xi_0^* + \Delta \xi^* \) is known as well as the plastic deformation variation.

\[
\{\Delta \xi^P\} = [D]^{-1} (\{\Delta \sigma^t\} - \{\Delta \sigma\})
\]

The plastic load \( \{\Delta F^P\} \) necessary to reequilibrate the structure is then given by

\[
\{\Delta F^P\} = \sum_m \int_{V_m} \left[ B_m \right]^T (\{\Delta \sigma_m^t\} - \{\Delta \sigma_m\}) \, dV_m
\]

This force is added to the second member of the equilibrium equation

\[
[K] \{\Delta U\} = \{\Delta R\} + \{\Delta F^P\}
\]

which is solved by external iterations.

1.2.3.4 Convergence criteria. An acceleration method

The external iterations are stopped when \( \{\Delta F^P\} \) does not change any more. That is done in the computer program by the following criteria. Let \( \sigma_n^* \) be the value of \( \sigma^* (\sigma_0 + \Delta \sigma_n^t - \Delta \sigma_n) \) and \( \sigma_n^* \) the value of \( \sigma^* (\sigma_0 + \Delta \sigma_n^t - \Delta \sigma_{n-1}) \). Then these iterations are stopped when

\[
P_n = \frac{|\sigma_n^* - \sigma_n^*|}{\sigma_n^*} \leq P_{\text{max}}
\]

for all the elements and where \( P_{\text{max}} \) is the convergence criteria supplied by the user.

The same method applies to the internal iterations but with \( P_{\text{max}} / 2 \) as convergence criteria.
Depending on material characteristics this convergence can be slow. An acceleration method based on the following extrapolation method was then included in the program.

Let $\Delta \varepsilon^*_n$ be the variation of equivalent plastic deformation obtained at the issue of the $n^{th}$ external iteration and the associated internal iterations. The rate of convergence $\tau_n$ is defined by

$$\tau_n = \frac{\Delta \varepsilon^*_n - \Delta \varepsilon^*_n-1}{\Delta \varepsilon^*_n-1 - \Delta \varepsilon^*_n-2}$$

This rate is supposed to stabilise after only a few iterations. At the user's demand every $m$ iterations the limit of the $\Delta \varepsilon^*_n$ sequence is then calculated by

$$\tau = \frac{\Delta \varepsilon^*_m - \Delta \varepsilon^*_m-1}{\Delta \varepsilon^*_m-1 - \Delta \varepsilon^*_m-2}$$

and

$$\Delta \varepsilon^*_b = \frac{1}{1-\tau} \left( \Delta \varepsilon^*_m - \tau \Delta \varepsilon^*_m-1 \right)$$

which is the limit of the geometric sequence. Then the plastic deformations are modified in the same proportions

$$\{\Delta \varepsilon^p\} = \frac{\Delta \varepsilon^*_b}{\Delta \varepsilon^*_m} \{\Delta \varepsilon^p\}$$

(The upper index $p$ is for plastic) and the corresponding plastic loads $\{\Delta \varepsilon^P\}$ are used for the following external iterations.

For some problems if this extrapolation method fails every few iterations the tangential stiffness matrix can be calculated at the user's demand.
1.2.3.4 Hardening material (Kinematic hardening)

The yield criteria is written $F(\sigma-a) = 0$ and the stress-strain curve supposed to be bilinear. The Prager-Ziegler hypothesis is used to determine the hardening parameter $\{a\}$.

\[ \{da\} = d\mu \{\sigma-a\} \]

$d\mu$ is obtained by differentiating the yield surface equation and using the plastic flow equations. That is

\[ \{d\varepsilon^p\} = d\varepsilon^{xp} \frac{\partial \varepsilon}{\partial \varepsilon} \{\sigma-a\} \]

\[ \sigma^* = f(\sigma-a) \]

\[ \frac{df}{d(\sigma-a)} \{\sigma-a\} = 0 \Rightarrow \left( \frac{df}{d(\sigma-a)} \right)^T \{\sigma-a\} = 0 \]

and

\[ \left( \frac{df}{d(\sigma-a)} \right)^T \{d\sigma\} = \left( \frac{df}{d(\sigma-a)} \right)^T \{da\} = \left( \frac{df}{d(\sigma-a)} \right)^T \{\sigma-a\} \] $d\mu$

Then

\[ d\mu = \left( \frac{df}{d(\sigma-a)} \right)^T \{d\sigma\} \]

But for constant $a$

\[ \left( \frac{df}{d(\sigma-a)} \right)^T \{d\sigma\} = d\sigma^* \]

and

\[ d\mu = \frac{d\sigma^*}{\sigma^*} = h \frac{d\varepsilon^{xp}}{\sigma^*} \]

where $h$ is the derivative of the tensile curve at $\varepsilon_0^*$. This modifies the algorithm previously described in the following manner.
\begin{align*}
\{d\sigma\}_1 &= \{d\sigma^t\}_1 - \begin{bmatrix} M \end{bmatrix} \frac{d\varepsilon^P}{\sigma} \{\sigma - \alpha\}_1 \\
\{da\}_1 &= \frac{k}{\sigma} \frac{d\varepsilon^P}{\sigma} \{\sigma - \alpha\}_1 \\
\{d(\sigma-\alpha)\}_1 &= \{d\sigma^t\}_1 - \begin{bmatrix} M \end{bmatrix} + h \begin{bmatrix} I \end{bmatrix} \{\sigma - \alpha\}_1 \frac{d\varepsilon^P}{\sigma}
\end{align*}

where \([I]\) is the unit matrix. It is seen that this equation is the equation (8) by changing \{\sigma-\alpha\} to \{\sigma\}. The same integration method can then be applied. This method gives simultaneously the solutions of the equation of plasticity and the displacements of the yield surface defined by \{da\}.

1.2.3.5 Application to creep problems

As usually the first step is to write the equation of equilibrium

\[ [B] \{\sigma\} = \{P\} \tag{1} \]

and the equation for the stresses

\[ \{\sigma\} = \{\sigma_0\} + [D] (\{\Delta\varepsilon^t\} - \{\Delta\varepsilon^F\}) \tag{2} \]

The creep flow is given by the following equation

\[ \{\Delta\varepsilon^F\} = \int_{t_o}^{t_o + \Delta t} \frac{2f}{\Xi(t_o)} \dot{\varepsilon}^P dt \tag{3} \]

on the time interval \([t_o, t_o + \Delta t]\).

Equations (1) and (2) are combined and as above it is obtained

\[ [K] [\Delta\varepsilon^P]_n = \{\Delta\varepsilon^P\} + [B] [D] (\{\Delta\varepsilon^P\}_0 + [B] [D] (\{\Delta\varepsilon^P\}_n)_{n-1} \tag{4} \]
and the same iterative method is applied. The question here is how do one obtain \( \{ \Delta e^p \} \) from (3). The integration is numerically conducted with the simple equation

\[
(\Delta e^p)^n = \frac{\Delta t}{2} \left[ (\varepsilon^*_{(\sigma)} t=t_0) + (\varepsilon^*_{(\sigma)} t=t_0+\Delta t) \right]
\]

and \( (\Delta e^p) \) is then put in equation (4). This last equation replaces the internal iterations previously defined. After equilibrium is obtained at the \( n^{th} \) iterate it is necessary to verify that the stress at the end of the time step is the stress which was used to determine \( (\Delta e^p) \). If it is not the case a new iteration is performed.

1.2.4 Material data input

For plastic or creep analyses the ability to introduce easily material data in various manners is very important. Generally these data are given point by point, sometimes an analytical curve is assumed and they are also cases where it is necessary to smooth and fit the data by numerical means. In the TRICO program the stress-strain curves are supplied by either two ways: point by point with the other data related to the problem or by a user supplied subroutine where any type of constitutive equation can be accounted for. The point by point data are used by TRICO in conjunction with a quadratic interpolation. The creep data are always given by the user supplied subroutine in the form of the function.

\[
\varepsilon^* = \mathcal{E}(\varepsilon^*, \sigma^*, t, T, \ldots, \ldots)
\]

where \( \varepsilon^* \) is the strain intensity, \( \sigma^* \) the stress intensity, \( t \) the time, \( T \) the temperature etc ... Any argument of this function can be omitted and other arguments can be added. If the creep data are given point by point the user supplied subroutine must be provided with an interpolation scheme. This way of dealing with creep leaves the designer free to choose any hardening rule (time hardening, strain hardening, or other rules).
1.2.5 **Input and output of TRICO**

a) **Input**

The input must always begin with some parameters on the number of nodes and elements and other values, to allocate the computer memory necessary for the problem. All the other data are input in free format fields in any order due to the use of code names to specify the different data categories. The input is generally composed by

- The mesh: coordinates of the points, numbers defining the elements
- The thicknesses of the elements
- Some material properties:
  - Young modulus
  - Poisson's ratio
  - Stress-strain curves (if given point by point)
- The loads
- The load and time increments
- The boundary conditions
- Parameters for non linear analysis: maximal number of iterations, convergence criteria
- Parameters for selection of the printed output.

b) **Output**

The results are listed or stored on computer files for latter retrieval by post-processor codes. They can be visualized on graphic display units, hard copy or plotters.
Fig 1.2.1 convergence criteria

\[ P_n = \frac{|B_n - A_n|}{|A_n|} < \tau_{\text{max}} \]
These post-processor codes can be used either by the batch processing mode or the interactive time sharing mode.

All input data are printed. Then for each time or load steps the following results are printed:

- number of plastic elements, actual convergence error, maximal stress and strain intensities. Actual loads and time increments. Number of iterations.
- At selected elements: the principal strains and curvature variations.

For selected time or load steps the following results are printed:

- The displacements and rotations
- The boundary reactions
- The principal stresses and moments

1.3 Elbow-Pipe Assembly Subjected to In-plane Moment Loading at 593°C

1.3.1 Problem description (ref. 6)

1.3.1.1 Geometry

The Elbow-pipe Assembly tested at Battelle-Columbus Laboratories is a 101.6 mm sched-10 90° elbow with a 152.4 mm bend radius welded to two 324 mm lengths of sched - 10 pipes. (Fig. 1.3.1). The elbow wall thicknesses and diameter were measured at selected grid points (table 1.3.1 and Fig. 1.3.2).
<table>
<thead>
<tr>
<th>α</th>
<th>Wall thicknesses in mm</th>
<th>Outside diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0°</td>
<td>90°</td>
</tr>
<tr>
<td>0°</td>
<td>3.02</td>
<td>3.2</td>
</tr>
<tr>
<td>22.5°</td>
<td>2.74</td>
<td>2.84</td>
</tr>
<tr>
<td>45°</td>
<td>2.76</td>
<td>2.54</td>
</tr>
<tr>
<td>67.5°</td>
<td>3.37</td>
<td>2.81</td>
</tr>
<tr>
<td>90°</td>
<td>2.79</td>
<td>3.07</td>
</tr>
</tbody>
</table>

Table 1.3.1 - Dimensions of elbow

The average wall thickness is 3 mm and average outside diameter is 114.53 mm. The pipe legs have a wall thickness of 3.07 ± 0.05 mm, an outside diameter of 114.25 ± 0.25 mm and a length of 323.85 mm.

A rigid frame shown on fig. 3.1 is used to apply the loads.

1.3.1.2 Loads and boundary conditions

As system of dead weights was used to load the assembly. These loads produce a constant moment in the assembly. On figure 1.3.3 the loads are noted as $F_1$ and $F_2$. The load histogram is shown on figure 1.3.4. A first load of 1382 N is applied on each loading point producing a bending moment of 843 Nm. The specimen is held at this load for 295 hours. The load is then increased to 1827 N producing a bending moment of 1114 Nm. This load is held for another 44 hours. At this time the test is ended. The temperature during all the test is maintained at 593°C ± 5.6°C.
The Elbow pipe assembly is rigidly fastened at the other end.

1.3.1.3 Material

The elbow and pipe legs are made of type 304 stainless steel. The material properties are given in reference 5.

Young modulus 149 652 N/mm²

Poisson's ratio 0.3

The stress-strain curve at 593°C is given by the following formula

\[ \varepsilon_p = 4.59 \times 10^{-4} \sigma^{9,377} \]

where \( \sigma \) is in psi and \( \varepsilon_p \) the plastic strain is in/in. The tensile curve values are given below in table 1.3.2 and the corresponding graphe is shown on fig. 1.3.5.

<table>
<thead>
<tr>
<th>( \sigma ) ksi</th>
<th>( \sigma ) (N/mm²)</th>
<th>( \varepsilon ) (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.5</td>
<td>44.8</td>
<td>0.0002996</td>
</tr>
<tr>
<td>8.5</td>
<td>58.7</td>
<td>0.0006</td>
</tr>
<tr>
<td>9</td>
<td>62.1</td>
<td>0.0008</td>
</tr>
<tr>
<td>10</td>
<td>69</td>
<td>0.0015</td>
</tr>
<tr>
<td>10.5</td>
<td>72.4</td>
<td>0.0021</td>
</tr>
<tr>
<td>11</td>
<td>75.8</td>
<td>0.0030</td>
</tr>
<tr>
<td>11.5</td>
<td>79.4</td>
<td>0.0043</td>
</tr>
<tr>
<td>12</td>
<td>82.8</td>
<td>0.0062</td>
</tr>
<tr>
<td>15</td>
<td>89.7</td>
<td>0.0125</td>
</tr>
<tr>
<td>17</td>
<td>117.2</td>
<td>0.1468</td>
</tr>
</tbody>
</table>

Table 1.3.2 - Tensile curve at 593°C
The creep data are given by the reference 5 table 2.

1.3.2 The finite element idealisation

1.3.2.1 Geometry

Half of the elbow pipe assembly due to symmetry conditions) is meshed with 770 triangular elements and 432 nodes. The mesh, the element numbering and nodal numbering are presented on figures 1.3.6, 1.3.7 and 1.3.8.

Each element is given its appropriate thickness. The equithickness lines are represented on figure 1.3.9.

The frame at the free end was not meshed but represented by a rigid flange. This flange is represented in the mesh by the last row of elements (749 to 770) which were given a 45 mm thickness.

1.3.2.2 Loads and boundary conditions

The forces are introduced directly on the mesh on points number 421 and 432. They are given a value taking in account the effect of the beams length. That is

\[
P_1^* \times R = \frac{P_1}{2} \times 304.8 \text{ mm}
\]

where \( R \) is the mean radius of the pipe, and, due to the symmetry condition, only half of the load is applied.

\[
P_1^* = \frac{1382 \times 304.8}{2 \times 55.63} = 3\,786.03 \text{ N}
\]

The same value is given to \( F_2^* \).

By the same formula the second load is obtained as 5005.12N.
The boundary conditions as mentioned previously are symmetry conditions for the XOY plane and of the clamped type conditions for the fixed pipe end. This was imposed by setting to zero value all the degrees of freedom of nodes 1 to 12 and symmetry condition on nodes 13, 25, ..., 421 and 24, 36, ..., 432.

1.3.2.3 Material

The stress-strain curve was entered point by point with the values of table 1.3.2.

The creep data were replaced by a similar matrix but with strain rates. These rates were calculated from table 1.3.3 by taking the ratio between strain differences and time differences for a given stress level. (For example at a stress of 14 Ksi between the 50th and 100th hour the rate is $89 \times 10^{-7}$ in/in/hour). These rates were included in a user supplied subroutine where linear interpolation between stress levels was programmed.

1.3.2.4 Miscellaneous data

The maximum number of iterations is set to 23 and the convergence criteria $\tau_{\text{max}}$ to $10^{-2}$. The total number of time and load increments is 51 (Fig. 1.3.10). The first five steps are used to impose the initial load at constant time (no hold time). Then the following time table is used:

- between 0 and 10 hours 10 steps
- between 10 and 100 hours 10 steps
- between 100 and 295 hours 14 steps

Then the load is incremented in five steps at constant time and between the 295th and 339th five time steps are used. The unloading is made in two load steps at 339 hours.
1.3.3 Results

1.3.3.1 Displacements

Due to the idealisation some additional computation are necessary to provide typical displacements that are comparable to the set presented in the Benchmark problem. The quantities $\delta_1$, $\delta_2$, $\delta_3$, $\delta_y$ and $\theta$ as defined in reference 6 and on figure 1.3.11 are obtained by taking the mean values of $u_A$ and $v_A$, the displacements in the $x$ and $y$ directions respectively, of the points 425 and 426 of the mesh. The following formulas are applied

$$\delta_1 = v_A - 304.8 \theta_A$$
$$\delta_2 = - v_A - 304.8 \theta_A$$
$$\delta_3 = - 165.1 \theta_A + u_A$$
$$\delta_y \approx v_A$$
$$\theta = - \theta_A$$

The results are presented on table 1.3.3 and figures 1.3.12, 1.3.13, 1.3.14, 1.3.15, 1.3.16. The comparison with experimental results shows that these computations over estimate the displacement. This difference is supposed to come for the material data which do not seem to take in account the metallurgical charges of the elbow material which was rolled, welded and hot bended. Some complementary work was conducted with ASME material data. These results are presented in Part 3 of this report.
Tableau 1.3.3 - Calculated displacements

1.3.3.2 Strains

The computed strains are given for the strain gages located on figure 1.3.17. The table 1.3.4 gives this values in micro deformation in function of time. The number of the gage corresponds to the number on figure 1.3.17.
### Table 1.3.4 Calculated strains

#### 1.3.3.3 Stresses

The stress distribution is represented by the Von Mises stress intensity isocurves. A map of these isocurves is given at the beginning and at the end of each load. Figure 1.3.18 for the initial load, figure 1.3.19 after 295 h, figure 1.3.20 at 295 h for the second load and figure 1.3.21 after 44 hours of hold time for the second load. The same maps are shown in close up by figures 1.3.22 to 1.3.25.

<table>
<thead>
<tr>
<th>Step number</th>
<th>time (hours)</th>
<th>5C</th>
<th>6C</th>
<th>7L</th>
<th>10C</th>
<th>11L</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>-1369</td>
<td>452</td>
<td>194</td>
<td>-1721</td>
<td>-478</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-1412</td>
<td>467</td>
<td>202</td>
<td>-1766</td>
<td>-493</td>
</tr>
<tr>
<td>8</td>
<td>3</td>
<td>-1453</td>
<td>481</td>
<td>210</td>
<td>-1810</td>
<td>-507</td>
</tr>
<tr>
<td>10</td>
<td>5</td>
<td>-1483</td>
<td>491</td>
<td>216</td>
<td>-1843</td>
<td>-518</td>
</tr>
<tr>
<td>15</td>
<td>10</td>
<td>-1536</td>
<td>510</td>
<td>227</td>
<td>-2028</td>
<td>-522</td>
</tr>
<tr>
<td>17</td>
<td>28</td>
<td>-1642</td>
<td>547</td>
<td>247</td>
<td>-2102</td>
<td>-578</td>
</tr>
<tr>
<td>20</td>
<td>55</td>
<td>-1734</td>
<td>577</td>
<td>262</td>
<td>-2142</td>
<td>-615</td>
</tr>
<tr>
<td>25</td>
<td>100</td>
<td>-1820</td>
<td>606</td>
<td>277</td>
<td>-2254</td>
<td>-651</td>
</tr>
<tr>
<td>30</td>
<td>169.6</td>
<td>-1915.5</td>
<td>638</td>
<td>295</td>
<td>-2369.5</td>
<td>-689</td>
</tr>
<tr>
<td>35</td>
<td>239</td>
<td>-1990</td>
<td>664</td>
<td>309</td>
<td>-2462</td>
<td>-719</td>
</tr>
<tr>
<td>39</td>
<td>295</td>
<td>-2033</td>
<td>679</td>
<td>317</td>
<td>-2517</td>
<td>-737</td>
</tr>
<tr>
<td>44</td>
<td>297</td>
<td>-3880</td>
<td>1186</td>
<td>542</td>
<td>-4959</td>
<td>-1520</td>
</tr>
<tr>
<td>46</td>
<td>312.6</td>
<td>-3906</td>
<td>1195</td>
<td>548</td>
<td>-4988</td>
<td>-1530</td>
</tr>
<tr>
<td>49</td>
<td>339</td>
<td>-3946</td>
<td>1208</td>
<td>556</td>
<td>-5033.5</td>
<td>-1546</td>
</tr>
<tr>
<td>51</td>
<td>339</td>
<td>-3030</td>
<td>887.1</td>
<td>387</td>
<td>-4025</td>
<td>-1336</td>
</tr>
</tbody>
</table>


13.4 Conclusions

The results presented for the Elbow-pipe assembly benchmark problem are very different from the measured values as given in reference 6. The deflection curves and strain curves in function of time have the same shape but the end values differ by a factor of 2 on total displacement (or a factor of 3 on creep displacement). Does the material data used in the computation correspond to the elbow material which was subjected to some metallurgical charges during the elbow manufacturing?
Fig. 1.3.1 Geometry
Fig 1.3.2 Location of grid points of table 1
Fig 1.3.3 loading system
Fig. 1.3.4 Load histogram
Fig. 1-3.5 Initial elastic-plastic loading curve for type 304 stainless steel (Hot 972796) at 593°C (1100°F).

Slope = \(149.6 \times 10^3\) MPa
\((21.7 \times 10^6\) psi)
Fig. 1.3.7. - Element numbering.
Fig. 1.3.8. - Node numbering.
Fig 1.3.10 Load and time increments

steps numbers are in brackets

[10 steps] [10 steps] [14 steps]
[5] [15] [25] [39] [44] [49]

f

[50] [51]

10 100 295 339

time (hours)
Fig. 1.3.11 interpretation of computer results
Fig 1.3.12 $\delta_1$ displacement
Fig. 1.3.13 $\delta$, displacement

TRICO

Experiment
Fig. 13.14. $\delta_3$ displacement
Fig 1.3.16

Rotation (radian)

Theta

Trico

Experiment

Theta rotation

Time (hours)
Strain Gage Length = 0.375"

C = Circumferential
L = Longitudinal

Gages are on outside surface

Fig. 13.37 High-temperature strain-gage locations.
Fig. 1.3.19. - Stress intensities isocurves. After 295 hours.
Fig. 1.3.20. - Stress intensities isocurves. Second load application at 295 hours.
Fig. 1.3.21. - Stress intensities isocurves. After 339 hours.
Fig. 1.3.23. - Close up of Fig. 1.3.19.
Fig. 1.3.24. - Close up of Fig. 1.3.20.
1.4 Elevated-Temperature Elastic-Plastic-Creep Test of an Elbow Subjected to In-Plane Moment Loading (Ref.7)

1.4.1 Problem description

1.4.1.1 Geometry

This assembly was a 304.8 mm, sched-20, 90° elbow and pipe fabricated from type 304 stainless steel. The geometry is presented on figure 1.4.1. The elbow and pipes wall thicknesses were measured and supplied in addenda 1 to reference 7. These values are reproduced in table 1.4.1. The positions of the grid points for the thickness measurement are given on fig. 1.4.2. The pipe numbers are defined on figure 1.4.1.

<table>
<thead>
<tr>
<th>Pipe number</th>
<th>Outside diameter (mm)</th>
<th>Wall thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>318.5</td>
<td>6.5</td>
</tr>
<tr>
<td>2</td>
<td>318.5</td>
<td>6.5</td>
</tr>
<tr>
<td>3</td>
<td>318.5</td>
<td>10.0</td>
</tr>
</tbody>
</table>

Table 1.4.1 (a)

The length of pipe 1 is 742.8 mm, the length of pipe 2 and 3 are supposed to be respectively 678.4 mm and 864 mm.

1.4.1.2 Loads and boundary conditions.

A transverse concentrated force is applied at one end of the assembly producing an in-plane bending moment in the elbow.

The load history is shown on figure 1.4.3. Seven loads were applied, each load being followed by a period of hold time. For steps 1 to 5 the load was removed at the end of the creep period. For steps 6, the load was increased directly into step 7.
The temperature was maintained to 600° C during all the test period which lasted 330 hours. The other end of the assembly is rigidly constrained.

1.4.1.3 Material.

The elbow and pipes are made of type 304 stainless steel. The material properties are given in reference 7.

Young modulus 15 300 Kg/mm²
Poisson’s ratio 0.3

The stress-strain curve at 600° C is given by the following formula:

\[ \varepsilon_p = 6.806 \times 10^{-10} \sigma^{8.045} \]

where \( \varepsilon_p \) is the plastic strain in percent and \( \sigma \) the stress in Kg/mm².

The creep data are given by the following equations.

\[ \varepsilon_c = \varepsilon_x (1-e^{-st}) + \varepsilon_t (1-e^{-rt}) + \varepsilon_m t \]

\( s = 4.233 \times 10^{-3} \) \( (\sinh (0.1001 \sigma))^{3.5} \)

\( r = 0.1 s \)

\( \varepsilon_m = 2.32 \times 10^{-4} \) \( (\sinh (6.901 \times 10^{-2} \sigma))^{6} \)

\( \varepsilon_t = 5 \varepsilon_m / r \)

where the strain units are percents and the stress units are Kg/mm². The time is in hours.

Supporting materials properties data were not available for the specific material used to fabricate the elbow. Properties
available in the literature were used in this benchmark.

1.4.2 The finite element idealisation

1.4.2.1 Geometry

Due to the symmetry only the half of the assembly is meshed with 680 elements and 385 nodes. The mesh, the element numbering and nodal numbering are given respectively on figures 1.4.4, 1.4.5 and 1.4.6. The rigid beam used to apply the loads is represented by a row of elements, numbered 661 to 680, whose thickness is set to 100 mm. The iso thickness curves in the elbow are given on figures 1.4.7 and 1.4.8.

1.4.2.2 Loads and boundary conditions.

The forces are applied on nodal point number 385 (figure 1.4.6)

The boundary conditions are symmetry conditions for the X0Y plane and of the clamped type condition for the fixed pipe end. All degrees of freedom of nodes 1 to 11 are set to zero value. The symmetry applies to nodes number 12, 23, ..., 375 and 22, 33, ..., 385. (figure 1.4.6.)

1.4.2.3 Material representation

The stress strain curve is entered point by point. The values are given in table 1.4.2.

<table>
<thead>
<tr>
<th>stress (Kg/mm²)</th>
<th>strain (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>0.00046</td>
</tr>
<tr>
<td>8</td>
<td>0.000648</td>
</tr>
<tr>
<td>9</td>
<td>0.000912</td>
</tr>
<tr>
<td>10</td>
<td>0.0014</td>
</tr>
<tr>
<td>11</td>
<td>0.0023</td>
</tr>
<tr>
<td>12.5</td>
<td>0.00536</td>
</tr>
<tr>
<td>14.5</td>
<td>0.016</td>
</tr>
<tr>
<td>16</td>
<td>0.034</td>
</tr>
</tbody>
</table>

Table 1.4.2. - Stress-strain curve.
The creep data were supplied, after differentiation of $\varepsilon_c$, via the subroutine already described in 1.2.4.

1.4.2.4 Miscellaneous data.

The maximum number of iterations was set to 28 and the convergence criteria to $10^{-2}$. The total number of time and load steps was 51. The following time table is used:

<table>
<thead>
<tr>
<th>step number</th>
<th>time (hours)</th>
<th>load (Kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>615</td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>615</td>
</tr>
<tr>
<td>3</td>
<td>25</td>
<td>615</td>
</tr>
<tr>
<td>4</td>
<td>45</td>
<td>615</td>
</tr>
<tr>
<td>5</td>
<td>96</td>
<td>615</td>
</tr>
<tr>
<td>6</td>
<td>96</td>
<td>0</td>
</tr>
<tr>
<td>7</td>
<td>96</td>
<td>853</td>
</tr>
<tr>
<td>8</td>
<td>126</td>
<td>853</td>
</tr>
<tr>
<td>9</td>
<td>166.5</td>
<td>853</td>
</tr>
<tr>
<td>10</td>
<td>166.5</td>
<td>0</td>
</tr>
<tr>
<td>11</td>
<td>166.5</td>
<td>993</td>
</tr>
<tr>
<td>12</td>
<td>200</td>
<td>993</td>
</tr>
<tr>
<td>13</td>
<td>236</td>
<td>993</td>
</tr>
<tr>
<td>14</td>
<td>236</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>236</td>
<td>1200</td>
</tr>
<tr>
<td>16</td>
<td>280</td>
<td>1200</td>
</tr>
<tr>
<td>17</td>
<td>304.5</td>
<td>1200</td>
</tr>
<tr>
<td>18</td>
<td>304.5</td>
<td>0</td>
</tr>
<tr>
<td>19</td>
<td>304.5</td>
<td>1270</td>
</tr>
<tr>
<td>20</td>
<td>324</td>
<td>1270</td>
</tr>
<tr>
<td>21</td>
<td>324</td>
<td>0</td>
</tr>
<tr>
<td>22</td>
<td>324</td>
<td>1335</td>
</tr>
<tr>
<td>23</td>
<td>325.5</td>
<td>1335</td>
</tr>
<tr>
<td>24</td>
<td>325.5</td>
<td>1382</td>
</tr>
<tr>
<td>25</td>
<td>330</td>
<td>1382</td>
</tr>
</tbody>
</table>

Table 1.4.3. Loading time table.
In this table, linear interpolation on loads or time increments is used between the given step numbers.
Figure 1.4.9 gives a schematic representation of this time table.

1.4.3 Results

1.4.3.1 Deflection

Table 1.4.4 gives the calculated deflections for all load steps. These values are graphically displayed on figure 1.4.10 and 1.4.11. Figure 1.4.10 compares the measured and computed values. As it is seen the calculations do not agree with measurements. Due to the sensitivity of the elasto-plastic models to material data and to the fact that for this Benchmark measured data are not available no conclusions can be drawn from the comparison. Figure 1.4.11 compares the same values but with a time-deflection curve.

1.4.3.2 Strains

Figure 1.4.12 (a) and (b) compares the strains obtained by calculation in the Benchmark Problem and by the TRICO program. As before no conclusion can be drawn.

1.4.4 Conclusions

The results obtained with the TRICO computer program are very far away from the measured values. Due to the hypothesis made on material data it seems than another Benchmark Test of this type is necessary to assess the validity or the invalidity of the constitutive equation for this case.
<table>
<thead>
<tr>
<th>Step number</th>
<th>Load (kg)</th>
<th>Time (hour)</th>
<th>Free end deflection (mm)</th>
<th>radial displacement at $\theta = 270^\circ$, $\phi = 0^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>615</td>
<td>0</td>
<td>16.33</td>
<td>1.062</td>
</tr>
<tr>
<td>2</td>
<td>615</td>
<td>5</td>
<td>16.53</td>
<td>1.080</td>
</tr>
<tr>
<td>3</td>
<td>615</td>
<td>10</td>
<td>16.71</td>
<td>1.096</td>
</tr>
<tr>
<td>5</td>
<td>615</td>
<td>25</td>
<td>17.19</td>
<td>1.139</td>
</tr>
<tr>
<td>7</td>
<td>615</td>
<td>45</td>
<td>17.72</td>
<td>1.185</td>
</tr>
<tr>
<td>11</td>
<td>615</td>
<td>96</td>
<td>18.76</td>
<td>1.275</td>
</tr>
<tr>
<td>12</td>
<td>0</td>
<td>96</td>
<td>2.76</td>
<td>0.247</td>
</tr>
<tr>
<td>14</td>
<td>853</td>
<td>96</td>
<td>25.70</td>
<td>1.740</td>
</tr>
<tr>
<td>17</td>
<td>853</td>
<td>126</td>
<td>29.12</td>
<td>2.018</td>
</tr>
<tr>
<td>20</td>
<td>853</td>
<td>166.5</td>
<td>32.59</td>
<td>2.292</td>
</tr>
<tr>
<td>21</td>
<td>0</td>
<td>166.5</td>
<td>10.40</td>
<td>0.866</td>
</tr>
<tr>
<td>23</td>
<td>993</td>
<td>166.5</td>
<td>36.23</td>
<td>2.527</td>
</tr>
<tr>
<td>25</td>
<td>993</td>
<td>188.8</td>
<td>39.53</td>
<td>2.778</td>
</tr>
<tr>
<td>28</td>
<td>993</td>
<td>236</td>
<td>45.34</td>
<td>3.213</td>
</tr>
<tr>
<td>29</td>
<td>0</td>
<td>236</td>
<td>19.51</td>
<td>1.553</td>
</tr>
<tr>
<td>32</td>
<td>1200</td>
<td>236</td>
<td>53.03</td>
<td>3.770</td>
</tr>
<tr>
<td>33</td>
<td>1200</td>
<td>250.7</td>
<td>55.75</td>
<td>3.955</td>
</tr>
<tr>
<td>35</td>
<td>1200</td>
<td>280</td>
<td>60.62</td>
<td>4.285</td>
</tr>
<tr>
<td>37</td>
<td>1200</td>
<td>304.5</td>
<td>64.26</td>
<td>4.532</td>
</tr>
<tr>
<td>38</td>
<td>0</td>
<td>304.5</td>
<td>33.04</td>
<td>2.525</td>
</tr>
<tr>
<td>40</td>
<td>1270</td>
<td>304.5</td>
<td>68.10</td>
<td>4.842</td>
</tr>
<tr>
<td>42</td>
<td>1270</td>
<td>314.3</td>
<td>69.64</td>
<td>4.943</td>
</tr>
<tr>
<td>44</td>
<td>1270</td>
<td>324</td>
<td>71.13</td>
<td>5.041</td>
</tr>
<tr>
<td>45</td>
<td>0</td>
<td>324</td>
<td>38.09</td>
<td>2.918</td>
</tr>
<tr>
<td>47</td>
<td>1335</td>
<td>324</td>
<td>76.01</td>
<td>5.428</td>
</tr>
<tr>
<td>48</td>
<td>1335</td>
<td>325.5</td>
<td>76.26</td>
<td>5.444</td>
</tr>
<tr>
<td>49</td>
<td>1382</td>
<td>325.5</td>
<td>81.67</td>
<td>5.862</td>
</tr>
<tr>
<td>51</td>
<td>1382</td>
<td>330</td>
<td>82.47</td>
<td>5.914</td>
</tr>
</tbody>
</table>

Table 1.4.4. - Calculated deflections.
Fig. 1.4.1. GEOMETRY
Wall Thickness Distribution in the Elbow Prior to Testing

<table>
<thead>
<tr>
<th></th>
<th>$\theta=0^\circ$</th>
<th>$\theta=30^\circ$</th>
<th>$\theta=60^\circ$</th>
<th>$\theta=90^\circ$</th>
<th>$\theta=120^\circ$</th>
<th>$\theta=150^\circ$</th>
<th>$\theta=180^\circ$</th>
<th>$\theta=210^\circ$</th>
<th>$\theta=240^\circ$</th>
<th>$\theta=270^\circ$</th>
<th>$\theta=300^\circ$</th>
<th>$\theta=330^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d=+45^\circ$</td>
<td>8.2</td>
<td>8.2</td>
<td>8.2</td>
<td>8.4</td>
<td>7.9</td>
<td>8.0</td>
<td>8.0</td>
<td>7.6</td>
<td>8.0</td>
<td>8.1</td>
<td>8.0</td>
<td>8.3</td>
</tr>
<tr>
<td>$d=+30^\circ$</td>
<td>7.5</td>
<td>7.5</td>
<td>7.4</td>
<td>7.7</td>
<td>7.8</td>
<td>7.9</td>
<td>7.4</td>
<td>7.8</td>
<td>7.8</td>
<td>7.6</td>
<td>7.4</td>
<td>7.4</td>
</tr>
<tr>
<td>$d=+15^\circ$</td>
<td>7.2</td>
<td>7.2</td>
<td>7.4</td>
<td>7.8</td>
<td>7.8</td>
<td>7.8</td>
<td>7.2</td>
<td>7.8</td>
<td>7.6</td>
<td>7.4</td>
<td>7.2</td>
<td>7.1</td>
</tr>
<tr>
<td>$d=0^\circ$</td>
<td>7.0</td>
<td>6.9</td>
<td>7.1</td>
<td>7.2</td>
<td>7.6</td>
<td>7.8</td>
<td>7.2</td>
<td>7.8</td>
<td>7.6</td>
<td>7.2</td>
<td>6.9</td>
<td>7.0</td>
</tr>
<tr>
<td>$d=-15^\circ$</td>
<td>6.8</td>
<td>7.0</td>
<td>7.2</td>
<td>7.4</td>
<td>7.7</td>
<td>7.5</td>
<td>7.4</td>
<td>8.0</td>
<td>7.4</td>
<td>7.3</td>
<td>7.2</td>
<td>7.0</td>
</tr>
<tr>
<td>$d=-30^\circ$</td>
<td>7.4</td>
<td>7.5</td>
<td>7.4</td>
<td>7.8</td>
<td>7.7</td>
<td>7.6</td>
<td>7.2</td>
<td>7.4</td>
<td>7.4</td>
<td>7.1</td>
<td>7.3</td>
<td>7.2</td>
</tr>
<tr>
<td>$d=-45^\circ$</td>
<td>8.0</td>
<td>8.4</td>
<td>8.0</td>
<td>8.0</td>
<td>7.8</td>
<td>7.6</td>
<td>7.6</td>
<td>7.5</td>
<td>8.0</td>
<td>8.0</td>
<td>8.0</td>
<td>8.2</td>
</tr>
</tbody>
</table>

Outside Diameter: 318.5 mm

Fig. 1.4.2. WALL THICKNESS MEASUREMENT POINTS
Fig. 1.4.3 Loading history
Fig. 1.4.5. NODAL NUMBERING
WALL THICKNESS DISTRIBUTION IN THE ELBOW
ISO-COURSES

Fig 1.4.7 ISO-THICKNESSES
Fig 1.4.9 Loading time table
Fig 1.4.10  LOAD-DEFLECTION CURVE
Fig 1.4.11  Deflection in time
Fig 1.4.12 (a). Radial displacement and circumferential strain distribution for plane $\phi = 0^\circ$ at the beginning of load step 3.

Fig 1.4.12 (b). Radial displacement and circumferential strain distribution for plane $\phi = 0^\circ$ at the end of load step 3.
1.5 - Room-Temperature Elastic-Plastic Response of a Thin Walled Elbow Subjected to in Plane Bending Loads

1.5.1 - Problem description (Ref. 17)

1.5.1.1 - Geometry

A thin-walled large diameter elbow was tested at CEA/DEMT at room temperature. It is a 180° elbow with a 762 mm bend radius, its thickness is 12 mm and external diameter 507 mm. Measurements of wall thickness were taken at prescribed grid points on the specimen, as shown in figure 2 of reference 17. Two pipe legs are welded to the elbow. Addendum 3 of reference 17 gives the thicknesses of these pipes. They are shown again here on figure 1.5.1.

1.5.1.2 - Loads and boundary conditions

The loads are applied at room temperature. The in plane bending moment is applied by hydraulic jacks pulling on a rigid structure which acts on the pipe legs. The loads are symmetrically applied. They are given by figure 3 and table 2 of reference 17.

1.5.1.3 - Material

The elbow and pipe legs are made of type 304 stainless steel. The material properties are given in reference 17.

Young modulus 191 300 N/mm²
Poisson's ratio 0.3
The stress strain curve at 20°C is given schematically on figure 5 of reference 17. Table 1.5.1 gives the corresponding values.

<table>
<thead>
<tr>
<th>$\sigma$ (N/mm$^2$)</th>
<th>$\varepsilon$ (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>0.00052</td>
</tr>
<tr>
<td>192</td>
<td>0.00123</td>
</tr>
<tr>
<td>228.7</td>
<td>0.00316</td>
</tr>
<tr>
<td>233</td>
<td>0.00404</td>
</tr>
<tr>
<td>300</td>
<td>0.0238</td>
</tr>
<tr>
<td>348</td>
<td>0.0595</td>
</tr>
</tbody>
</table>

Table 1.5.1 - Tensile curve at room temperature

1.5.2 - The finite element idealisation

1.5.2.1 - Geometry

Only one fourth of the structure is meshed with 504 elements and 286 nodes. Figures 1.5.2, 1.5.3 and 1.5.4 show respectively the mesh, the element numbering and node numbering.

Each element is given an appropriate thickness. The elbow thickness is set to the nominal value of 12 mm (figure 1.5.1)

1.5.2.2 - Loads and boundary conditions

The mesh does not include the sleeves on which acted actually the hydraulic jacks during the tests. The applied
forces are imposed by the following method. A force $f_1$ and a couple $f_2$ are combined as shown on figure 1.5.5.

The couple $f_2$ is determined by the following equation

$$f_2 \times d = f_1 \times l$$

where $d$ is the pipe mean diameter (500 mm) and $l$ is the sleeves length (388 mm). $f_1$ is the force corresponding to the nominal moment values given in the benchmark problem data (The arm length is 1 900 mm).

The boundary conditions are symmetry conditions about the XOY plane and the YOZ plane at the top of the elbow (figure 1.5.2). Some of the nodal displacements are restrained to avoid rigid body motion. These conditions applied to the mesh are summarized in table 1.5.2.

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symmetry XOY</td>
<td>1, 14, ..., 274</td>
</tr>
<tr>
<td>Symmetry XOY</td>
<td>9, 22, ..., 286</td>
</tr>
<tr>
<td>Symmetry YOZ</td>
<td>275, 276, ..., 285</td>
</tr>
<tr>
<td>Fixed x direction displacement</td>
<td>274, 286</td>
</tr>
<tr>
<td>Fixed y direction displacement</td>
<td>274, 286</td>
</tr>
<tr>
<td>Fixed rotation around z axis</td>
<td>274 and 286</td>
</tr>
</tbody>
</table>

Table 1.5.2 - Boundary conditions

The loads applied actually in the computation where modified in three occasions.
As it is seen on the experimental results a little load increment produces a large deflection in the curved part of the load-deflection curve. This is also reflected by the stress-strain curve trend. It is also known that the deflection measurements are more accurate than the force measurements which does integrated other effects as friction for example. Based on these considerations it was decided to modify the applied loads as given in reference 17. These modifications were intended to simulate a deflection controlled load application. In any case they were limited to a maximum of 5%.

Due to this trial and error method the pressure load step was difficult to implement in the loading sequence. As it is observed this pressure load step has not a very important effect on the following cycles. As a consequence the pressure load was deleted from the three dimensional shell analysis.

The third modification in the loading history was to consider two loading sequence history parts. The first is represented by the loading sequence given by load numbers 19, 20, 22, 24, 26, 27, ..., 41 minus the pressure load. The second loading sequence given by load numbers 19, 40, 42, 44, 46, ... 58 with two unloadings after load numbers 48 and 54. This choice does not take into account loads 1 to 19 because on the experimental results it is seen that the elbow's response is proportional to the loading. For loads 19 to 58 the separation in two loading histories was due to the lack of one of the program's features. To do these calculations it is necessary to use a large displacements theory and the computer program was not ready to account for large displacements in connection with kinematic hardening (a memory allocation problem). So the first loading history sequence was computed with cyclic loading, a bilinear stress-strain curve and kinematic hardening but without geometrical nonlinearities. The second loading history sequence was computed with the tensile curve given point by point, isotropic hardening, monotonous loading and geometrical nonlinearities.
1.5.2.3 - Material

a) Bilinear representation for the cyclic loading history

A bilinear representation of the stress strain curve is necessary with kinematic hardening. The chosen curve is given by table 1.5.3.

<table>
<thead>
<tr>
<th>$\sigma$ (N/mm$^2$)</th>
<th>$\varepsilon$ (mm/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>212</td>
<td>0.000109</td>
</tr>
<tr>
<td>258</td>
<td>0.01</td>
</tr>
</tbody>
</table>

Table 1.5.3 - Bilinear stress-strain curve

b) Stress-strain curve for the monotonous loading history

The points of table 1.5.1. are entered with the other data relevant to this problem.

1.5.3 - Description of the implementation of large displacement in the $\epsilon^*$-Hoffmann method

The procedure employed in the TRICO program for this computation was not described in 1.2 and the large displacements calculations in conjunction with the $\epsilon^*$-Hoffmann for structural non linear analysis is given.
1.5.3.1 - The fundamental hypothesis

For shell analysis the equation of static equilibrium is written on the deformed body by assuming that for local axis related to the element the stresses are equal on the initial and deformed bodies. If the initial structure is noted \( C_0 \) and the deformed structure \( C \) then the forces, displacements and stresses are respectively

\[
\begin{align*}
F_0 & \quad \text{and} \quad F = F_0 + \Delta F \\
u_0 & \quad \text{and} \quad u = u_0 + \Delta u \\
\sigma_0 & \quad \text{and} \quad \sigma = \sigma_0 + \Delta \sigma
\end{align*}
\]

The equation of equilibrium is written

\[
[B] T \{ \sigma \} = \{ F \}
\]

with the same notations as in 1.2.

This produces the following expression

\[
[B_0] T \{ \Delta \sigma \} + [B-B_0] T \{ \sigma \} = \{ \Delta F \}
\]

where \([B_0]\) is the \([B]\) matrix defined on the initial structure. This last equation is to be solved.

1.5.3.2 - Resolution of the equilibrium equation

The basic equation which is solved for all non linear problems in the CEASEMT system can be expressed in the form of

\[
[A] (\Delta X)_n = (\Delta F) + [B_0]^T \{ \sigma_0 \} - [B]^T \{ \sigma \}_{n-1} + [A] (\Delta X)_{n-1}
\]

where \( A \) can be either of the following matrices
- The classical stiffness matrix $K$
- $K + K_G$ ($K_G$ the geometrical stiffness matrix)
- $K_T + K_G$ ($K_T$ the tangential stiffness matrix)

and $X$ any variable.

The choice between these matrices is relative to the type of problem which has to be solved. The convergence rate is greatly influenced by such a choice.

It must be observed that the initial stress method is implied and that the forces due to the stresses are to be computed. This is what was already described in 1.2 for the plasticity part of the $\varepsilon^*$-Hoffmann method.

To be more specific some special cases will be discussed.

a) Elasticity

The $(\Delta \sigma)$ stress increment is related to the deformations calculated on $C_0$ by

$$\{\Delta \sigma\} = \{D\} \{\Delta \varepsilon\} = \{D\} \{\Delta \varepsilon\}^I + [D] \{\Delta \varepsilon\}^II$$

where $\{\Delta \varepsilon\}^I$ and $\{\Delta \varepsilon\}^II$ are respectively the first and second order strains. This leads to the iterative equation

$$[K] \{\Delta U\}_n = \{\Delta F\} - [R-B^T] \{\sigma\}_{n-1} - [B^T] \{\Delta \varepsilon\}^II_{n-1}$$

b) Plasticity

This time $(\Delta \sigma)$ is written as
\{\Delta \sigma\} = [D] \{\Delta \varepsilon\}_{\text{Total}} - [D] \{\Delta \varepsilon\}^P

and the analytical integration procedure of 1.2 is applied. This leads to the following iterative equation

\[ [K] \{\Delta U\}_n = \{\Delta F\} - [B - B_0]^T \{\sigma\}_{n-1} + [B_0]^T [D] \{\Delta \varepsilon\}^P_{n-1} \]

\[ - [B_0] [D] \{\Delta \varepsilon\}^\Pi_{n-1} \]

To accelerate convergence it can be useful to add the tangential matrices \(K_T\) or \(K_G\) or both.

This can be written, for example, as:

\[ [K + K_G] \{\Delta U\}_n = \{\Delta F\} - [B - B_0]^T \{\sigma\}_{n-1} + [B_0]^T [D] \{\Delta \varepsilon\}^P_{n-1} \]

\[ - [B_0] [D] \{\Delta \varepsilon\}^\Pi_{n-1} + [K_G] \{\Delta U\}_{n-1} \]

1.5.4 - Results

1.5.4.1 - Load-deflection curve for the cyclic loading history

A table is given below for the load-deflection curve calculated with the data presented in 1.5.2. The first column gives the load step number used for the calculation. The second column gives the corresponding load number of Table 2 of Reference 17. The third column gives the actually applied moments and the fourth column the computed deflection. These values are to be compared to the measured ones given in the fifth column. The last column gives the percentage of incrementation which was applied on the nominal load value given in the benchmark problem of Reference 17.
<table>
<thead>
<tr>
<th>Computer program load number</th>
<th>Bench. problem load number</th>
<th>Moment (10^6 \text{ Nm})</th>
<th>Calculated deflect. (mm)</th>
<th>Measured deflect. (mm)</th>
<th>Load increase (%)</th>
<th>Observations</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>19</td>
<td>0</td>
<td>0</td>
<td>-0.7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>20</td>
<td>9.76</td>
<td>10.507</td>
<td>10.3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>22</td>
<td>11.62</td>
<td>12.514</td>
<td>13.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>24</td>
<td>14.18</td>
<td>15.725</td>
<td>18.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>26</td>
<td>17.65</td>
<td>22.545</td>
<td>24.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4(b)</td>
<td></td>
<td>18.08</td>
<td>23.787</td>
<td></td>
<td>+2.5</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>27</td>
<td>0</td>
<td>4.324</td>
<td>8.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>28</td>
<td>-7.43</td>
<td>-3.924</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>29</td>
<td>-14.86</td>
<td>-15.584</td>
<td>-17.8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>30</td>
<td>-15.23</td>
<td>-16.426</td>
<td>+2.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>31</td>
<td>-15.61</td>
<td>-17.250</td>
<td>+5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>32</td>
<td>0</td>
<td>0.454</td>
<td>-2.3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>13.93</td>
<td>16.03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td>15.89</td>
<td>19.34</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td>17.85</td>
<td>23.351</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td>19.82</td>
<td>30.637</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td>21.78</td>
<td>44.172</td>
<td>23.7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td>32</td>
<td>20.72</td>
<td>5.3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td>33</td>
<td>0</td>
<td>(not computed)</td>
<td>-3.60</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td>35</td>
<td>-17.84</td>
<td>-17.016</td>
<td>-5</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 1.5.4 - Load-deflection values for the cyclic loading history.
Figure 1.5.6 represents graphically these results and the experimental values are shown on it for comparison. It is obvious than the results are unsatisfactory. This is due to the non availability of the geometrical nonlinearity capability for this case, especially for the closing loads. (The program will be modified and this case rerun later). For the last loading steps in opening (load number 39, 40 and 41) the differences can be explained by the difference observed for load number 38 which influences the next cycle. As was mentioned before the pressure has no great effects on following cycles. The main effect is the difference in behaviour of elbows in opening and closing.

1.5.4.2 - Strains results for the cyclic loading history

For comparison with experimental data the computed strains are averaged between adjacent element. The comparison is made for load number 24 with an applied moment of 176 580 mN. The method of averaging will be described with the help of figure 1.5.7. Their the two rows of elements situated on each side of the line of strain gages are shown. The shaded triangles are those participating in the averaging process. The strain gages are supposed to be situated in the middle of the common side of the two triangles which are used for each gage. For example take the strain gages at the measurement point number 12. This measurement point is supposed to be in the middle between nodal points 209 and 210. The strain values corresponding to this measurement point are then the average of the strain values computed on triangles number 385 and 362. This procedure is applied for all measurements points.

Table 1.5.5 presents the results for the computed strains. The first line is the measurement point the second line gives the triangle numbers used for averaging. The third line gives the circumferential strains corresponding to each triangle.
<table>
<thead>
<tr>
<th>Measurement point number</th>
<th>12</th>
<th>11</th>
<th>10</th>
<th>9</th>
<th>8</th>
<th>7</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>2</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Triangle number</td>
<td>385</td>
<td>388</td>
<td>389</td>
<td>392</td>
<td>393</td>
<td>396</td>
<td>397</td>
<td>400</td>
<td>401</td>
<td>404</td>
<td>405</td>
<td>408</td>
</tr>
<tr>
<td></td>
<td>362</td>
<td>363</td>
<td>366</td>
<td>367</td>
<td>370</td>
<td>371</td>
<td>374</td>
<td>375</td>
<td>378</td>
<td>379</td>
<td>382</td>
<td>383</td>
</tr>
<tr>
<td>Circumferential strains</td>
<td>438</td>
<td>663</td>
<td>886</td>
<td>377</td>
<td>-698</td>
<td>-2360</td>
<td>-1220</td>
<td>312</td>
<td>866</td>
<td>772</td>
<td>448</td>
<td>241</td>
</tr>
<tr>
<td></td>
<td>418</td>
<td>626</td>
<td>1126</td>
<td>332</td>
<td>-829</td>
<td>-2417</td>
<td>-1978</td>
<td>158</td>
<td>861</td>
<td>734</td>
<td>411</td>
<td>245</td>
</tr>
<tr>
<td>Longitudinal strain</td>
<td>160</td>
<td>307</td>
<td>332</td>
<td>1593</td>
<td>1383</td>
<td>-178</td>
<td>-299</td>
<td>-1207</td>
<td>-1172</td>
<td>-278</td>
<td>-188</td>
<td>-18</td>
</tr>
<tr>
<td></td>
<td>141</td>
<td>183</td>
<td>527</td>
<td>932</td>
<td>1141</td>
<td>1050</td>
<td>-255</td>
<td>-1145</td>
<td>-592</td>
<td>-542</td>
<td>-72</td>
<td>-45</td>
</tr>
<tr>
<td>Average circumferential strains</td>
<td>428</td>
<td>644.5</td>
<td>1006</td>
<td>354.5</td>
<td>-763.5</td>
<td>-2388</td>
<td>-1599</td>
<td>235</td>
<td>864</td>
<td>753</td>
<td>430</td>
<td>243</td>
</tr>
<tr>
<td>Average longitudinal strain</td>
<td>150</td>
<td>245</td>
<td>440</td>
<td>1263</td>
<td>1262</td>
<td>436</td>
<td>-277</td>
<td>-1176</td>
<td>-882</td>
<td>-410</td>
<td>-130</td>
<td>-32</td>
</tr>
</tbody>
</table>

Table 1.5.5 - Computed strains and averaged values in $10^{-6}$ mm/mm (load number 26)
The fourth line the longitudinal strains corresponding to each triangle. Lines five and six are the average circumferential and longitudinal strains.

These averaged values are used to produce the curves of figure 1.5.8. and 1.5.9 respectively for the circumferential and longitudinal strains. The measured values are given for comparison on the same figures. It is seen that a good correlation is obtained. It must be remarked that this comparison is for load number 26 when the large effects of the elbow's closing are not present. The load deflection value at this point is also near the measurement point.

1.5.4.3 - Load-deflection curve for the monotonic loading history

These results correspond to the second part of the loading as described in 1.5.2.2. Isotropic hardening is used in conjunction of the large displacement option of TRICO. The loads are applied monotonically.

The table 1.5.6 and figure 1.5.10 represent the load-deflection curve calculated by the TRICO computer code. As explained earlier the comparison must be based on the deflection, due to the accuracy of its measurement. It is seen that for a given deflection the calculated load is within 10% of the measured one.

Also the curve trend shows that the geometrical nonlinearities are important in this problem and are well taken in account by the computer code.

This non-linear behavior is also seen if the three linear loadings are compared that is to say the initial loading and the two loadings following the unloadings (load numbers 48
and 54). These three lines are not parallel due to the geometrical changes of the elbow. The calculated behavior follows very nearly the measured one and reflect this phenomena. This is due to the use of the "change of geometry at each step" option.

1.5.4.4 - Strains for the monotonic loading history

The strains for load number 46 are given. Table 1.5.7 gives the circferential and longitudinal strains for the triangles represented on figure 1.5.7. These values are averaged, the average values are given by the two last rows of table 1.5.7. The average values are shown on figures 1.5.11 and 1.5.12. They are compared to the measured values.
<table>
<thead>
<tr>
<th>Measurement point number</th>
<th>12</th>
<th>11</th>
<th>10</th>
<th>9</th>
<th>8</th>
<th>7</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>2</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Triangle numbers</td>
<td>385</td>
<td>388</td>
<td>389</td>
<td>392</td>
<td>393</td>
<td>396</td>
<td>397</td>
<td>400</td>
<td>401</td>
<td>404</td>
<td>405</td>
<td>408</td>
</tr>
<tr>
<td></td>
<td>392</td>
<td>363</td>
<td>366</td>
<td>367</td>
<td>370</td>
<td>371</td>
<td>374</td>
<td>375</td>
<td>378</td>
<td>379</td>
<td>382</td>
<td>383</td>
</tr>
<tr>
<td>Circumferential strains</td>
<td>1088</td>
<td>2057</td>
<td>5001</td>
<td>3882</td>
<td>514</td>
<td>-6811</td>
<td>-2648</td>
<td>4558</td>
<td>6261</td>
<td>3829</td>
<td>1969</td>
<td>956</td>
</tr>
<tr>
<td></td>
<td>871</td>
<td>1782</td>
<td>5372</td>
<td>5094</td>
<td>948</td>
<td>-6322</td>
<td>-5052</td>
<td>4221</td>
<td>5287</td>
<td>4031</td>
<td>1594</td>
<td>932</td>
</tr>
<tr>
<td>Longitudinal strain</td>
<td>1912</td>
<td>2234</td>
<td>2052</td>
<td>6323</td>
<td>5703</td>
<td>168</td>
<td>-248</td>
<td>-3819</td>
<td>-3937</td>
<td>-893</td>
<td>-670</td>
<td>57</td>
</tr>
<tr>
<td></td>
<td>1944</td>
<td>2015</td>
<td>3752</td>
<td>4000</td>
<td>4599</td>
<td>4628</td>
<td>370</td>
<td>-2752</td>
<td>-1595</td>
<td>-1577</td>
<td>100</td>
<td>90</td>
</tr>
<tr>
<td>Average circumferential strain</td>
<td>980</td>
<td>1920</td>
<td>5186</td>
<td>4488</td>
<td>731</td>
<td>-6566</td>
<td>-3850</td>
<td>4390</td>
<td>5774</td>
<td>3930</td>
<td>1782</td>
<td>944</td>
</tr>
<tr>
<td>Average longitudinal strain</td>
<td>1913</td>
<td>2125</td>
<td>2902</td>
<td>5162</td>
<td>5151</td>
<td>2398</td>
<td>61</td>
<td>-3285</td>
<td>-2766</td>
<td>-1235</td>
<td>-285</td>
<td>74</td>
</tr>
</tbody>
</table>

Table 1.5.7 - Computed strains and averaged values in $10^{-6}$ mm/mm (load number 46).
Fig 1.5.1 Geometry
Fig 1.5.2 - Mesh
Fig 1.5.3 Nodal numbering
Fig 1.5.4 - Element numbering
Fig 1.5.5. Load application
Fig. 1.5.6 Load-deflection curve (cyclic Loading history)

Couple $10^4$Nm

Experimental value
Calculated value

Load numbers are given in table 2 of Ref. 17
Fig 1.5.7 Triangles used for averaging
Fig. 1.5.8
CIRCONFERENTIAL STRAINS
LOADING MOMENT: 176580 mN

- Calculated strains
○ Experimental results

Nodal points

Measurement points

0.001
0.0001

12 11 10 9 8 7 6 5 4 3 2 1 0
209 210 211 212 213 214 215 216 217 218 219 220 221

0.00265
Fig. 1.5.9

AXIAL STRAINS
LOADING MOMENT: 176580mN

- Calculated strains
○ Experimental results

measurement points

nodal points

0.001

0.001

209 210 211 212 213 214 215 216 217 218 219 220 221
Fig 1.5.10 Elbow load-deflection behavior for load steps 42 to 59.
Fig 1.5.11 Circumferential strain. Load 46. Moment = 313 920 mN

Fig 1.5.12 Longitudinal strain. Load 46. Moment = 313 920 mN
2.1 - Introduction

The TEDEL piping and beam structures analysis program is part of the CEASEMT finite element system. A simplified piping analysis method was developed at CEA/DEMT and included in this program for elastic-plastic-creep static or dynamic analysis.

An elbow element with geometric nonlinearity capability was developed for the same type of analysis. This element was used for the piping benchmark problem calculations.

The COCO pre-processor previously described received specialised modules for piping or beam structures mesh generation. The post-processors TEMPS and ESPACE can be used as well as with any program of the system.

2.2 - Description of the simplified piping analysis computer code TEDEL (ref. 13)

2.2.1 - General features

The code TEDEL is based on the finite element method. The usual elements are beams, straight pipes and curved pipes. Three other "elements" are also available: a beam element with any cross-section (the stiffness matrix must be supplied by the
user), a 90° miter elbow and a non-reinforced branch connection element. The action of fluids on the solid structure can be taken in account. The straight elements have two nodes with six degrees of freedom each. The curved elements are made of a given small number of straight elements and all the internal degrees of freedom are taken in account in the calculation.

The TEDEL program can solve static or dynamic problems for linear or nonlinear material or geometrical behaviour. The solving capability is limited only by the out of core storage limits of the computer due to the dynamic storage allocation.

The loading and boundary conditions possibilities are extensive. It is possible to apply concentrated forces or moments, dead weight, distributed forces, accelerations or thermal loads. Displacements can be imposed in any direction. Symmetry conditions, relations between displacements, or between forces and displacements, as well as pin joints between elements can be imposed.

2.2.2 - The global method applied to pipes

The analysis in the plastic-creep range in TEDEL takes advantage of the global method (reference 9). Here the generalized point is the cross-section, and its coordinates are those of the neutral axis of the pipe. It is then possible to use a beam type global method for nonlinear analysis. These methods are presented extensively in references 10 and 11.

The generalised stresses are the tension, $N$ the bending $M_b$ the torsion $M_t$ and the pressure $P$. These quantities are normalised for practical reasons and this produces the following expressions:
Normal stress  \[ \sigma_n = \frac{N}{\pi D t} \]
Pressure stress  \[ \sigma_p = \frac{P D}{2 t} \]
Bending stress  \[ \sigma_b = \frac{4M_b}{\pi D^2 t} \]
Torsion stress  \[ \sigma_t = \frac{4M_t}{\pi D^2 t} \]

(D is the pipe diameter and \( t \) its thickness).

To these "stresses" correspond generalised strains

\[ \epsilon_n, \epsilon_p, \epsilon_b, \epsilon_t \]

defined by the work variation. This variation is given by

\[ \delta W = \sigma_n \delta \epsilon_n + \sigma_p \delta \epsilon_p + \sigma_b \delta \epsilon_b + \sigma_t \delta \epsilon_t \]

thus the generalised strains can be geometrically defined by the following relations

Length variation  \[ \epsilon_n = \frac{A l}{l} \]
Diameter variation  \[ \epsilon_p = \frac{A D}{D} \]
Curvature variation  \[ \epsilon_b = \frac{D}{\lambda} \chi \]
Torsional variation  \[ \epsilon_t = \frac{D}{\lambda} \phi \]

were \( \chi \) is the curvature of the neutral axis and \( \phi \) the torsion of the pipe.

When only bending and torsion are important (this is generally the case in high temperature components) the number of useful unknowns reduces to the two couples \((\sigma_b, \epsilon_b)\) and \((\sigma_t, \epsilon_t)\).
The yield surface, of the Von Mises type, is then defined by

\[ \sigma_y^2 = a_b^2 \sigma_b^2 + a_t^2 \sigma_t^2 \]

where \( a_b \) and \( a_t \) are coefficients depending of the collapse load of the pipe. In TEDEL these coefficients are taken as

\[ a_t = \sqrt{3/2} \]
\[ a_b = \frac{\pi}{4} \]

for straight parts. For elbows the procedure is described in the paragraph 2.2.3. Having defined the yield surface the plastic flow is derived from HILL's normaly rule with

\[ \epsilon_b = \lambda \frac{\partial F}{\partial \sigma_b} = a_b^2 \frac{\sigma_b}{\sigma_y} \, d\epsilon^* \]
\[ \sigma_t = \lambda \frac{\partial F}{\partial \sigma_t} = a_t^2 \frac{\sigma_t}{\sigma_y} \, d\epsilon^* \]

The value of \( d\epsilon^* \) is obtained by the hypothesis that the stress-strain curve relates the equivalent stress and strain intensities. This procedure is valid for isotropic hardening.

For kinematic hardening \( \sigma_b \) and \( \sigma_t \) are replaced by \( \sigma_b - \delta_b \) and \( \sigma_t - \delta_t \) where \( \delta_b \) and \( \delta_t \) depend on the material work hardening caracteristics. They are obtained by the Prager-Ziegler rule

\[ d\delta_b = (\sigma_b - \delta_b) \, d\nu \]
\[ d\delta_t = (\sigma_t - \delta_t) \, d\nu \]
where $d_u$ is given by the following relationship

$$d_u = \frac{d\varepsilon_b}{d\sigma_y} \frac{d\sigma_b}{\sigma_y} + \frac{d\varepsilon_t}{d\sigma_y} \frac{d\sigma_t}{\sigma_y}$$

This relation is obtained from the total differential of the yield surface with constant yield stress and replacing $d\sigma_b$, $d\sigma_t$, $\frac{\partial F}{\partial \sigma_b}$ and $\frac{\partial F}{\partial \sigma_t}$ by their values.

The $\varepsilon^*$-Hoffmann procedure already described in the first part for the TRICO program is used to solve the equilibrium and plasticity equations by external and internal iterations.

### 2.2.3 The elbow element

The TEDEL elbow element is defined by its curvature radius and flexibility and stress concentration factors.

The element itself is geometrically defined by a limited number of straight pipe elements. The intermediate nodal points are used in the computation of displacements. A numerical integration along the neutral axis replaces the well known analytical formulas for curved pipe or beams. This idealisation is very simple to introduce in the TEDEL program and was of great use when large displacements, taking into account the changes of cross section in elbows, were introduced for this element.

The plastic behaviour of elbows is based on its collapse load. For this purpose the results of reference 12, which provide an upper and a lower bound can be used. They are reproduced on figure 2.2.1 with some experimental results of reference 15. From this it is possible to obtain a conservative value of $\alpha_b$ for elbows as a function of the geometrical parameter

$$\lambda = \frac{Rt}{r^2}$$
here $R$ is the radius of curvature, $t$ the thickness and $r$ the mean radius of the elbow. The following relations is taken in TEDEL for the elbow element

$$a_b = \frac{2\pi}{3} \lambda^{-2/3} \quad \text{(but not less than } \frac{1}{4})$$

The value of $a_t$ is unchanged because torsion does not have significant effect on the variations of the cross section's shape.

The flexibility coefficient is taken equal to $1.65/\lambda$.

Having defined these parameters the calculation takes in account the changes in cross section's shape and inertia in the following way. The equation of equilibrium on the deformed structure is written as

$$BEI(x) \chi = \mathbf{F}$$

where $B$ is the linear operator relating forces and stresses (the stiffness matrix $K$ is equal to $\int_{\text{volume}} B^T DB \, d(\text{volume})$ with $D$ matrix of material characteristics), $E$ Young modulus, and $I$ the moment of inertia depending of the curvature symbolises the external forces. This nonlinear equation is solved by iterations with the following scheme

$$B \{EI_0 \chi\}_n = \mathbf{F} + B \{EI_0 \chi\}_{n-1} - B \{EI(x) \chi\}_n$$

These iterations are imbedded in the external and internal iterations previously described. The modified inertia is taken in account for the bending stress calculation by the formula

$$a_b = a \frac{M R}{I}$$

where $a$ is a dimensionless shape factor.
The strains in each cross section of the elbow are obtained under some supplementary hypotheses (Discussion of reference 15). The TEDEL code gives global results at nodal points. Between nodes a linear variation is supposed for the curvative $\chi$ and bending moment $M_b$. Thus these two quantities are known at any cross section.

The deformation depends on the radial displacement $W$ (figure 2.2.2). The $W$ displacement is developed in its Fourier's series which is truncated to the two first terms

$$W = a (\cos 2 \theta + b \cos 4 \theta)$$

The circular extension is neglected and the $W$ displacement can be written as

$$W = \frac{dV}{d\theta}$$

where $V$ is the tangential displacement.

Thus $V = -a \left( \frac{\sin 2 \theta}{2} + b \frac{\sin 4 \theta}{4} \right)$.

The perpendicular displacement, to the neutral axis of the beam or the pipe, is then given by

$$u = V \cos \theta + W \sin \theta$$

and the axial strain $\varepsilon_1$ is deduced from the well known formula

$$\varepsilon_1 = \frac{u + x \frac{R_y}{y + R}}{y + R}$$

where $R$ is the radius of curvature and $y = r \sin \theta$ (r mean radius of the elbow).

The circumferential strain $\varepsilon_\theta$ is linear through the thickness and expressed by
\[ \varepsilon_\theta = \frac{1}{r^2} \left( \frac{d^2 W}{d\theta^2} + W \right) x \quad \text{with} \quad x \in \left[ -\frac{t}{2}, \frac{t}{2} \right] \]

Replacing \( W \) by its value one obtains

\[ \varepsilon_\theta = - \frac{3a}{r^2} \left( \cos 2\theta + 5b \cos 4\theta \right) x \]

The two constants \( a \) and \( b \) are left to be evaluated. For engineering purposes \( b \) is taken as 0.2 and \( a \) can be deduced from the maximal value of the circumferential deformation given by the code TEDEL.

2.2.4 - Other features

The incremental algorithm as described in paragraph 1.2.3 of part one is used by the TEDEL program. The convergence criteria is the same as in TRICO. The material data input is also the same as in TRICO. In fact the same subroutine, without any change, was used for both calculations.

2.2.5 - Input and output of TEDEL

a) Input

The input must begin with some parameters on the number of nodes and elements or other necessary data to allocate the computer memory. The other data are input in free format fields in any order due to the use of code names for distinguishing the different data categories. The input stream is composed by

- The mesh
- The element thicknesses
- The relevant data for elbows: radius of curvature, orientation, number of straight parts and their thickness
- Some material properties (Young modulus, Poisson's ratio, tensile curves if given point by point)
- The loads
- The time and load increments
- The boundary conditions
- The list of the elements for which plastic deformation output is necessary.
- Parameters for non-linear analysis: number of iterations, convergence criteria.

b) Output

For each load or time step the program prints:

- The number of iterations
- The number of plastic elements
- For each node: three displacements and three rotations
- For each restrained nodal displacement: the reaction
- For each element and for both nodes the torsional moment on x, the flexural moment on y, the shear forces on y and z where x is in the element direction and y and z are the orthogonal directions. The longitudinal stress is also given and computed by dividing the tension by the cross section surface
- For each element: the stress intensities defined above and the cumulated plastic strain intensities
- For a selected number of elements: the maximal circumferential and axial strains.
Fig 2.2.1 Collapse load of elbows

Fig 2.2.2 Elbow cross section
2.3 - Elbow-pipe Assembly Subjected to in Plane Moment Loading at 593°C (ref. 6)

This problem is fully described in 1.3 and only the idealisation for the simplified piping method computer program TEDEL will be discussed here.

2.3.1 - Finite element idealisation

The geometry of figure 1.3.1 is discretised in 14 elements and 15 nodes. The elbow is made with 9 of these elements (figure 2.3.1).

Each element is given an appropriate thickness. Due to the non uniform thickness at a cross-section of the elbow the following formula is used to define an equivalent constant thickness with the same bonding inertia for the cross-section:

\[ t = \frac{1}{\pi} \int_{-\pi}^{\pi} t(\theta) \sin^2 \theta \, d\theta \]

where \( t(\theta) \) is the thickness of the elbow in function of the angle \( \theta \) and \( t \) is the equivalent thickness. Table 2.3.1 gives the thicknesses attributed to each element.
<table>
<thead>
<tr>
<th>Element number</th>
<th>Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.305</td>
</tr>
<tr>
<td>2</td>
<td>0.3095</td>
</tr>
<tr>
<td>3</td>
<td>0.3111</td>
</tr>
<tr>
<td>4</td>
<td>0.3124</td>
</tr>
<tr>
<td>5</td>
<td>0.3074</td>
</tr>
<tr>
<td>6</td>
<td>0.3023</td>
</tr>
<tr>
<td>7</td>
<td>0.3068</td>
</tr>
<tr>
<td>8</td>
<td>0.3112</td>
</tr>
<tr>
<td>9</td>
<td>0.3091</td>
</tr>
<tr>
<td>10</td>
<td>0.3062</td>
</tr>
<tr>
<td>11</td>
<td>0.305</td>
</tr>
<tr>
<td>12</td>
<td>0.3057</td>
</tr>
<tr>
<td>13</td>
<td>0.3057</td>
</tr>
<tr>
<td>14</td>
<td>0.3057</td>
</tr>
</tbody>
</table>

Table 2.3.1 - Element thicknesses.

The inertia of elements 12, 13 and 14, representing the rigid frame, was amplified by setting their mean radius to 150 mm. The other elements have a mean radius of 57.15 mm.

The loads are imposed on points 14 and 15.

The boundary condition is applied by setting all degrees of freedom of node 1 to zero.

The stress-strain curve is entered point by point and the creep data are given by the same subroutine as in TRICO and already described in 1.3.2.3. For the sake of completeness
a second creep data subroutine was used with the analytical creep data given in addendum 2 to reference 5. The two calculations are compared.

The maximal number of iterations is set to 50 and the convergence criteria to $10^{-2}$. The total number of time and load increments is 51. The time table is represented on figure 2.3.2.

2.3.2 - Results

a) Creep data in matrix form.

2.3.2.1 - Displacements

The displacements are directly given on the mesh points with the following correspondence

$$
\delta_1 = v_{14} \\
\delta_2 = -v_{15} \\
\delta_3 = u_{13} \\
\theta = \theta_{12}
$$

where $v$ is the displacement in the $y$ direction and $u$ is the displacement in the $x$ direction. The indexes are nodal point numbers.

Table 2.3.2 gives the displacements at a selected number of time steps. Figures 2.3.3, 2.3.4, 2.3.5, 2.3.6 and 2.3.7 give respectively the values of $\delta_1$, $\delta_2$, $\delta_3$, $\delta_y$, $\theta$ in function of the time. The values are compared to the experimental ones.
### Table 2.3.2 - Calculated displacement.

<table>
<thead>
<tr>
<th>Step number</th>
<th>Time (hours)</th>
<th>( \delta_1 ) (mm)</th>
<th>( \delta_2 ) (mm)</th>
<th>( \delta_3 ) (mm)</th>
<th>( \delta_y ) (mm)</th>
<th>( \theta ) ((10^{-2} \text{ rad}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>12.443</td>
<td>8.330</td>
<td>19.688</td>
<td>2.056</td>
<td>3.404</td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>12.952</td>
<td>8.666</td>
<td>20.489</td>
<td>2.143</td>
<td>3.543</td>
</tr>
<tr>
<td>10</td>
<td>5</td>
<td>13.307</td>
<td>8.899</td>
<td>21.045</td>
<td>2.204</td>
<td>3.639</td>
</tr>
<tr>
<td>17</td>
<td>28</td>
<td>14.506</td>
<td>9.687</td>
<td>22.924</td>
<td>2.410</td>
<td>3.965</td>
</tr>
<tr>
<td>20</td>
<td>55</td>
<td>15.282</td>
<td>10.215</td>
<td>24.167</td>
<td>2.533</td>
<td>4.179</td>
</tr>
<tr>
<td>25</td>
<td>100</td>
<td>16.029</td>
<td>10.720</td>
<td>25.360</td>
<td>2.654</td>
<td>4.384</td>
</tr>
<tr>
<td>30</td>
<td>169.6</td>
<td>16.768</td>
<td>11.216</td>
<td>26.532</td>
<td>2.776</td>
<td>4.587</td>
</tr>
<tr>
<td>61</td>
<td>312.6</td>
<td>41.482</td>
<td>28.140</td>
<td>66.423</td>
<td>6.671</td>
<td>11.416</td>
</tr>
<tr>
<td>66</td>
<td>339</td>
<td>36.253</td>
<td>25.009</td>
<td>58.712</td>
<td>5.622</td>
<td>10.050</td>
</tr>
</tbody>
</table>
2.3.2.2 - Strains

The computed strains are given for the cross-sections corresponding to the strain gages used in the experiments. These results are shown with the corresponding experimental results on figures 2.3.8 to 2.3.22. The observed differences can be explained, as will be done in part three, by the stress-strain curve used in this benchmark calculation.

b) Analytical creep data

2.3.2.3 - Displacements

Due to the low cost of the simplified method it was possible to do a second calculation with an analytical representation of the creep data. The obtained displacements are given in table 2.3.3. It can be seen than the difference between the two calculations are not very important at the end of the test. But at 295 h (at the end of the first load) the analytical creep data produces more computed creep.
<table>
<thead>
<tr>
<th>Step number</th>
<th>Time (hours)</th>
<th>$\delta_1$ (mm)</th>
<th>$\delta_2$ (mm)</th>
<th>$\delta_3$ (mm)</th>
<th>$\delta_y$ (mm)</th>
<th>$\theta$ $(10^{-2}$ rd)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>12.443</td>
<td>8.330</td>
<td>19.688</td>
<td>2.056</td>
<td>3.404</td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>12.993</td>
<td>8.695</td>
<td>20.557</td>
<td>2.149</td>
<td>3.554</td>
</tr>
<tr>
<td>15</td>
<td>10</td>
<td>13.726</td>
<td>9.171</td>
<td>21.697</td>
<td>2.278</td>
<td>3.752</td>
</tr>
<tr>
<td>20</td>
<td>55</td>
<td>16.005</td>
<td>10.709</td>
<td>25.333</td>
<td>2.648</td>
<td>4.381</td>
</tr>
<tr>
<td>25</td>
<td>100</td>
<td>17.203</td>
<td>11.510</td>
<td>27.235</td>
<td>2.847</td>
<td>4.706</td>
</tr>
<tr>
<td>30</td>
<td>169.6</td>
<td>18.158</td>
<td>12.136</td>
<td>28.731</td>
<td>3.010</td>
<td>4.966</td>
</tr>
<tr>
<td>39</td>
<td>295</td>
<td>18.993</td>
<td>12.675</td>
<td>20.037</td>
<td>3.159</td>
<td>5.194</td>
</tr>
<tr>
<td>59</td>
<td>295</td>
<td>41.867</td>
<td>28.404</td>
<td>66.993</td>
<td>6.731</td>
<td>11.522</td>
</tr>
<tr>
<td>61</td>
<td>312.6</td>
<td>42.031</td>
<td>28.512</td>
<td>67.250</td>
<td>6.759</td>
<td>11.567</td>
</tr>
<tr>
<td>64</td>
<td>339</td>
<td>42.271</td>
<td>28.670</td>
<td>67.628</td>
<td>6.800</td>
<td>11.632</td>
</tr>
<tr>
<td>66</td>
<td>339</td>
<td>36.732</td>
<td>25.332</td>
<td>59.424</td>
<td>5.700</td>
<td>11.181</td>
</tr>
</tbody>
</table>

Table 2.3.3 - Calculated displacements
3.4 - Conclusions

The results presented for the elbow-pipe assembly and obtained with the simplified piping analysis computer code TEDEL differ very much from the measured values. They differ even more than those obtained with the TRICO computer code. No conclusions can be drawn here and the discussion is rejected to part three of this report. In fact in part three some of the observed discrepancies are analysed and the formulated hypothesis to explain them are verified.
Fig 2.3.2  load and time increments
Fig. 2.3.3

\[ \delta_1 (\text{mm}) \]

\[ \delta_1 \text{ displacement} \]

Experimental data for \( \delta_1 \) over time (hours)

- **Teddle**
Fig. 2.3.4

\[ \delta_2 \text{ (mm)} \]

---

\[ \delta_2 \text{ displacement} \]

---

time (hours)

---

Teddle

---

experiment

---

Fig. 2.3.4
Fig. 2.3.5

Graph showing \( \delta_3 (\text{mm}) \) displacement over time (hours). The graph includes two lines: one labeled "Tedel" and the other labeled "experiment."
Fig. 2.3.7

rotation (radian)

\( \vartheta \)

Tedel

experiment

\( \vartheta \) rotation

time (hours)

Fig. 2.3.7
Fig: 2.3.8 Longitudinal strains on the $\alpha=10^\circ$ plane at 0 hour.

Gage n°7
Load 843 mN
DEFORMATION CIRCONFÉRENTIELLE

Fig. 2.3.9. Circumferential strains on the $\alpha = 10^\circ$ plane at 0 hour
2.3.10. Longitudinal strains on the $\alpha = 10^\circ$ plane after 295 hours
Fig. 2.3.11. Circumferential strains on the $\alpha = 10^\circ$ plane after 295 hours
Fig. 2.3.12. Longitudinal strains on the $\alpha = 10^\circ$ plane after 295 hours
Fig. 2.3.13. Circumferential strains on the $\alpha = 10^\circ$ plane after 295 hours.
Fig. 2.3.14. Longitudinal strains on the $\alpha = 10^\circ$ plane after 339 hours
Fig. 2.3.15. Circumferential strains on the $\alpha = 10^\circ$ plane after 339 hours

\begin{itemize}
  \item $x$ Gage n°5
  \item $O$ Gage n°6
  \item Load 1114 mN
\end{itemize}
Fig. 2.3.16  Longitudinal strains on the $\alpha = 45^\circ$ plane at 0 hours
Fig. 2.3.17. Circumferential strains on the $\alpha=45^\circ$ plane at 0 hour
Fig 2.3.18. Longitudinal strains on the α = 45° plane after 295 hours

x Gage no 11
Load 843 mN
Fig. 2.3.19. Circumferential strains on the $\alpha = 45^\circ$ plane after 295 hours.
Fig. 2.3.20. Longitudinal strains on the $\alpha=45^\circ$ plane after 295 hours
Fig. 2.3.21. Circumferential strains on the $\alpha=45^\circ$ plane after 295 hours

$\text{Gage n° 10}$
$\text{Load 1114 mN}$
Fig. 2.3.22. Longitudinal strains on the $\alpha = 45^\circ$ plane after 339 hours

x Gage nº 11
Load 1114 mN
Fig. 2.3.23 Circumferential strains on the $\alpha = 45^\circ$ plane after 339 hours
2-4. Elevated - Temperature Elastic - Plastic - Creep Test of on Elbow Subjected to In Plane Moment Loading (Réf. 7) -

This problem is now solved with the simplified method.

2-4.1. Finite element idealisation -

The geometry of figure 1.4.1 is discretised with 19 elements and 20 nodes, 18 piping elements and one beam element. The elbow is made of 12 of these piping elements. This mesh is shown on figure 2.4.1.

Each piping element is given an appropriate thickness. The formula being the same as in 2.3.1. Table 2.4.1 gives the thicknesses given to each element.

<table>
<thead>
<tr>
<th>Element number</th>
<th>Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6.5</td>
</tr>
<tr>
<td>2</td>
<td>6.5</td>
</tr>
<tr>
<td>3</td>
<td>7.879</td>
</tr>
<tr>
<td>4</td>
<td>7.387</td>
</tr>
<tr>
<td>5</td>
<td>7.387</td>
</tr>
<tr>
<td>6</td>
<td>7.283</td>
</tr>
<tr>
<td>7</td>
<td>7.283</td>
</tr>
<tr>
<td>8</td>
<td>7.271</td>
</tr>
<tr>
<td>9</td>
<td>7.271</td>
</tr>
<tr>
<td>10</td>
<td>7.387</td>
</tr>
<tr>
<td>11</td>
<td>7.387</td>
</tr>
<tr>
<td>12</td>
<td>7.575</td>
</tr>
<tr>
<td>13</td>
<td>7.575</td>
</tr>
<tr>
<td>14</td>
<td>8.05</td>
</tr>
<tr>
<td>15</td>
<td>6.5</td>
</tr>
<tr>
<td>16</td>
<td>6.5</td>
</tr>
<tr>
<td>17</td>
<td>10.</td>
</tr>
<tr>
<td>18</td>
<td>10.</td>
</tr>
</tbody>
</table>

Table 2.4.1 - THICKNESSES
The beam element is a very rigid element and used only for the load application on point number 20.

The boundary condition are applied by setting to a zero value all the degrees of freedom of node number one.

The stress-strain curve is entered point by point and the creep data are introduced by the same subroutine which was used in the three dimensional computation and described in 1.4.2.3.

The maximal number of iterations is set to 23 and the convergence criteria to $10^{-3}$.

The total number of load and time increments is 141. The following time table is used:

<table>
<thead>
<tr>
<th>Step number</th>
<th>Time</th>
<th>Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>615</td>
</tr>
<tr>
<td>14</td>
<td>32</td>
<td>615</td>
</tr>
<tr>
<td>23</td>
<td>96</td>
<td>615</td>
</tr>
<tr>
<td>25</td>
<td>96</td>
<td>0</td>
</tr>
<tr>
<td>27</td>
<td>96</td>
<td>600</td>
</tr>
<tr>
<td>31</td>
<td>96</td>
<td>853</td>
</tr>
<tr>
<td>36</td>
<td>116</td>
<td>853</td>
</tr>
<tr>
<td>41</td>
<td>166.5</td>
<td>853</td>
</tr>
<tr>
<td>45</td>
<td>166.5</td>
<td>0</td>
</tr>
<tr>
<td>53</td>
<td>166.5</td>
<td>993</td>
</tr>
<tr>
<td>58</td>
<td>186.5</td>
<td>993</td>
</tr>
<tr>
<td>63</td>
<td>236</td>
<td>993</td>
</tr>
<tr>
<td>71</td>
<td>236</td>
<td>0</td>
</tr>
<tr>
<td>79</td>
<td>236</td>
<td>1200</td>
</tr>
<tr>
<td>84</td>
<td>256</td>
<td>1200</td>
</tr>
<tr>
<td>89</td>
<td>304.5</td>
<td>1200</td>
</tr>
<tr>
<td>97</td>
<td>304.5</td>
<td>0</td>
</tr>
</tbody>
</table>
2-4.2. Results -

2-4.2.1. Displacements -

Figure 2.4.2. gives the load-deflection curves of point 20 of the mesh. These results are compared to the measured ones. It is seen that the first load gives the same results but that for the consecutive ones the observed differences are significant. Table 2.4.3. gives numerical values of these displacements.

<table>
<thead>
<tr>
<th>Step number</th>
<th>Time (hours)</th>
<th>Displacement (mm)</th>
<th>Load (kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0</td>
<td>18.64</td>
<td>615</td>
</tr>
<tr>
<td>14</td>
<td>32</td>
<td>19.96</td>
<td>615</td>
</tr>
<tr>
<td>19</td>
<td>67.55</td>
<td>21.38</td>
<td>615</td>
</tr>
<tr>
<td>23</td>
<td>96</td>
<td>22.49</td>
<td>615</td>
</tr>
<tr>
<td>25</td>
<td>96</td>
<td>3.85</td>
<td>0</td>
</tr>
<tr>
<td>31</td>
<td>96</td>
<td>32.78</td>
<td>853</td>
</tr>
<tr>
<td>36</td>
<td>116</td>
<td>37.91</td>
<td>853</td>
</tr>
<tr>
<td>41</td>
<td>166.5</td>
<td>49.47</td>
<td>853</td>
</tr>
<tr>
<td>45</td>
<td>166.5</td>
<td>23.62</td>
<td>0</td>
</tr>
<tr>
<td>53</td>
<td>166.5</td>
<td>55.95</td>
<td>993</td>
</tr>
<tr>
<td>58</td>
<td>186.5</td>
<td>62.53</td>
<td>993</td>
</tr>
</tbody>
</table>

Table 2.4.2 - LOADING TIME TABLE
2-4.2.2. Strains

The calculated strains are compared to those calculated in the Benchmark. In fact in the Benchmark only the radial displacements are measured and the strains are deduced from these measures. Figure 2.4.4 gives the radial displacements at the beginning and at the end of load step number 3. The measured values are also represented. Figure 2.4.5 gives the corresponding circumferential strains calculated by TEDEL and the "measured values".

Table 2.4.3 - COMPUTED DEFLECTION VALUES

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>63</td>
<td>236</td>
<td>76.19</td>
<td>993</td>
</tr>
<tr>
<td>71</td>
<td>236</td>
<td>46.09</td>
<td>0</td>
</tr>
<tr>
<td>79</td>
<td>236</td>
<td>148.58</td>
<td>1200</td>
</tr>
<tr>
<td>84</td>
<td>256</td>
<td>154.42</td>
<td>1200</td>
</tr>
<tr>
<td>89</td>
<td>304.5</td>
<td>166.97</td>
<td>1200</td>
</tr>
<tr>
<td>97</td>
<td>304.5</td>
<td>130.59</td>
<td>0</td>
</tr>
<tr>
<td>105</td>
<td>304.5</td>
<td>200.1</td>
<td>1270</td>
</tr>
<tr>
<td>109</td>
<td>309.5</td>
<td>201.47</td>
<td>1270</td>
</tr>
<tr>
<td>113</td>
<td>324</td>
<td>205.34</td>
<td>1270</td>
</tr>
<tr>
<td>121</td>
<td>324</td>
<td>166.85</td>
<td>0</td>
</tr>
<tr>
<td>131</td>
<td>324</td>
<td>240.85</td>
<td>1335</td>
</tr>
<tr>
<td>134</td>
<td>325.5</td>
<td>241.29</td>
<td>1335</td>
</tr>
<tr>
<td>136</td>
<td>325.5</td>
<td>270.41</td>
<td>1382</td>
</tr>
<tr>
<td>141</td>
<td>330</td>
<td>271.78</td>
<td>1382</td>
</tr>
</tbody>
</table>

Figure 2.4.3 shows the same results but in function of the time. Experimental results are given on this figure for comparison.
2.4.1 Maillage du tuyau
Fig 2.4.2 LOAD DEFLECTION CURVE
Fig 2.4.3. Time-deflection curve

- Experimental
- TEDEL

TIME (Hours)
deflection (mm)
0 50 100 150 200 250 300 350 400
0 100 200 300 400
Fig. 2.4. Radial displacements at the beginning and at the end of load step number 3.
Fig. 2.4.5 Calculated circumferential strains at the beginning and at the end of load step number 3
2 -5. **Room Temperature Elastic - Plastic Response of a Thin Walled Elbow Subjected to In Plane Bending Loads (Simplified Method)**

2 -5.1. **Finite element idealisation**

2 -5.1.1. **Geometry**

The geometry of figure 2.5.1 is discretised with 17 elements and 12 nodes as shown on figure 2.5.1. The mesh comprises only the portion of the assembly above the measurement point shown on figure number 2 of addendum three to the International Piping Benchmark Problem compilation. This is due to the choice of a deflection controlled load application. The thickness of the elements is 12 mm.

2 -5.1.2. **Loads and boundary conditions**

The only boundary condition consists to apply symmetry conditions to node number 12. This is done by setting to a zero value the displacement in the x direction and the rotation around the z axis (z is perpendicular to the plane of the elbow).

As said above the loads are deflection controlled. After the experience gained with the three dimensional TRICO analysis it was decided to apply the measured deflection at the measurement point (point number 1 on the mesh) and to compute the moment at the top of the elbow. In fact this value is given by the reaction at node 12. The loading history is given by table 2.5.1. below. The loads are applied in the x direction and are in mm. Only half of the load is applied due to symmetry conditions.
Table 2.5.1 - LOADING HISTORY

A representation of the loading history is given on figure 2.5.2.

The pressure load is applied at step number 17. Its value is 2.01 N/mm². This affects the deflection and it is necessary to correct its value in proportion. The chosen deflection value is the value at Benchmark load number 30. At load number 33 the pressure is unloaded. For load numbers 41 to 58 the same procedure of deflection controlled loads is applied.
2-5.1.3. Material -

A multilayer hardening model is used to describe the material behavior. Two layers of bilinear material are used. This produces a trilinear stress-strain curve. This curve is shown on figure 2.5.3. in dotted lines. The mesh is duplicated and the computations are carried on 24 elements (12 for each material). The same thickness (12 mm) is given to both layers.

The two bilinear stress-strain curves whose summation produces the trilinear curve are defined by the following values:

- Material 1
  - Young modulus 13 700 N/mm$^2$
  - Tangential modulus 680 N/mm$^2$
  - Poisson's ratio 0.3

- Material 2
  - Young modulus 181 300 N/mm$^2$
  - Tangential modulus 680 N/mm$^2$
  - Poisson's ratio 0.3

The trilinear curve has then the following characteristics:

- Young modulus 195 000 N/mm$^2$
- First tangential modulus 14380 N/mm$^2$
- Second tangential modulus 1360 N/mm$^2$
2 -5.1.4. **Miscellaneous data** -

The maximal number of iterations is set to 27 and the convergence criteria to $3 \cdot 10^{-3}$.

2 -5.2. **Results** -

The figure 2.5.4. shows the calculated load deflection curve obtained with the TEDEL program. The comparison is made between the computed and measured bending moments. Good agreement is seen for the general behavior. The effects of internal pressure does not seem to be well taken in account but the changes of geometry are very well represented for this range of loadings. This is reflected by the good results obtained for the closing moments 28, 34 and 38.

Unfortunately this can not be said for loadings 42 to 58 where the geometrical changes are more important. Figure 2.5.5 gives the results obtained for loads 42 to 58 with the same input data as for load numbers 29 to 41. It is observed that the TEDEL program does not give a good agreement in this range. This is due to two causes:

- The effect of straight parts on the elbow is not taken in account in the TEDEL program (this accounts for the main part in the divergence with measured values).

- The change in inertia due to the change of the shape of the elbow's cross section is approximated only to its first order term.

Modifications of the program are now in progress to include the inertia calculated from the shape defined by the w function given in 2.2. up to the third order term. Then another development will include the straight parts effects on elbows.
2-5.3. Conclusion -

The room temperature tested elbow is analysed by a simplified method. The results are in good agreement for the loads which do not create large cross sectional shape deformations in elbows. For the larger loads the model must be modified to include a more precise formula for calculating the inertia in function of the curvature and to take in account the restraining effect of the straight parts.
Fig 2.5.1

MESH

76.2 mm
Fig 2.5.2
LOADING HISTORY

Deflection

Pressure Load

N
Fig 2.5.3 Tensile curve
LOAD-DEFLECTION CURVE

Couple in $10^4$ mN

Deflection in mm

Deflection in mm

Fig 2.5.4

30 → TEDEL Calculation
31 → Experimental Results
Fig 2.5.5
LOAD DEFLECTION CURVE
FOR LOADS N° 41 TO 58

Experimental values
* Calculated values
PART THREE

ANALYSIS OF BENCHMARK TESTS AND COMPUTER RESULTS

3. Evaluations of the benchmark problems

3.1. Introduction

The difficulties in solving the individual benchmark problems and the discrepancies between computed results and measured ones provided the ground for an analysis of the computer analyses and also of the benchmark tests themselves. Part one and two of this report intended to solve the proposed benchmark problems. The observed differences between calculations and measurements as well as between the three dimensional and the simplified analysis raised new questions. Another set of calculations were then performed and are described here.

The constitutive equations were described in part one and two for each program and are not discussed. The influence of the different ingredients used in conjunction with the constitutive equations are analysed. They are: the material data, the geometrical data and the influence of straight parts on elbows. The influence of finite element discretisation is not discussed here but one must bear in mind that the mesh size, the boundary condition, load application and material data representation has also some influence on the results.

A paragraph is devoted to the analysis of cost effectiveness of simplified methods compared to three dimensional calculations.

Improvements are suggested for the computer programs and especially for the simplified method. Also improvements and suggestions are recommended for additional benchmark problems on the same subjects.
3-2. **Influence of material data** -

3-2.1. *Elbow pipe assembly subjected to In-plane Moment Loading at 593°C (Battelle's Test)* -

3-2.1.1. **The tensile curve** -

As was observed in 1.3 and 2.3 the computed results are very different from the measured values of deflection. In fact the initial load produces about twice more deflection when computed than when measured.

The same benchmark problem was then computed but with a new tensile curve. The curve was taken in ASME code case 1592 for the 304 stainless steel at 1100°F (593°C). Figure 3.2.1 gives a comparison between the proposed curved in reference 5 and the code case curve.

With the CC1592 Tensile Curve, the results obtained with TEDEL and TRICO are depicted on figure 3.2.2 where the $\delta_1$ deflection is presented in function of the time. It is seen that the initial load gives now a computed results (for both programs) which is more or less equal to the measured value. For the second load application the elastic deflection is also well represented by both methods.

The stress-strain data given in reference 5 seem questionable for use in conjunction of this test. The elbow was plated then rolled, welded and hot bented. As a consequence the stress-strain curve can be modified.

3-2.1.2. **Creep data** -

It is also seen on figure 3.2.2 than for creep the results obtained by this new calculations are as before for away from the experimental values. The same questions as above for the tensile curve can be asked for the creep data.
To see the influence of material data on the constitutive equations another computation was made of this Benchmark Test with the material data of reference 5. The computation is done with the simplified piping analysis computer program TEDEL.

The results appear on fig. 3.2.3 where the two calculations are compared between themselves and with the measured strain values.
Fig 3.2.1 Comparison between tensile curves

N/mm²

C.C 1592

ref. 5

$\frac{\Delta L}{L}$

0 0.01 0.02 0.03
Fig 3.2.2 Time deflection curves calculated with the C.C.1592 tensile curve
Fig 3.2.3 Load deflection curve with Ref. 5 material data.

Benchmark material data

Ref. 5 material data
3-3. Influence of straight parts on elbows -

3-3.1. Introduction

It is not intended here to analyse the well known problem of the influence of straight parts on elbows as a phenomena. This effect is similar to the flange effects studied by G.E. Findlay and J. Spence in reference 22. The question we are interested in is: "given the same geometrical and material data why are the results obtained with the simplified method used in TEDEL so different from those obtained with the three dimensional shell analysis program TRICO". The hypothesis we made was that the main cause of this was that the simplified method did not yet account for the effects of straight parts on elbows. More calculations were needed to assess these ideas and this paragraph presents the results.

3-3.2. Elbow-Pipe Assembly subjected to In-Plane Moment Loading at 593°C

3-3.2.1. Radial displacement analysis -

The maximal radial displacement along the assembly is given in function of the curvilinear abscissa on figure 3.3.1., for both programs. (The computations are made with the CC 1592 tensile curve). It is seen on this figure that this displacement for TEDEL is constant for the elements of the elbow and equal to zero for the straight parts. This is not the case for the TRICO computation were the value of the maximal radial displacement varies constantly along the elbow and passes through a maximum at the middle of the elbow. It can also be observed that this maximum is smaller that the constant value given by TEDEL.
To demonstrate that these discrepancies are due to the straight parts effects on the elbow the original assembly was modified. The elbow is supposed to be the half part of a 180° elbow and calculations are made on the elbow with one pipe leg and symmetry conditions at the other end. Figure 3.3.2. gives the maximal radial displacement computed by both programs. It is seen than the straight part effect is less felt and that the value given by TEDEL is near the average value of the values given by TRICO.

This 180° geometry is computed again with the tensile curve of CC 1592 (fig. 3.2.1.) and creep data of reference 5. The horizontal displacement $\delta$ is given for both programs on figure 3.3.3 ($\delta$ is defined on the figure) and the global rotation of the assembly is given on figure 3.3.4. Good agreement between the two computations is seen. To give an idea of this last comparison for the original assembly with a 90° elbow and two pipe legs figure 3.3.5 presents the compared rotations obtained by both programs at the same point (point 12 of TEDEL'S mesh).

3-3.3. Elevated Temperature Elastic - Plastic - Creep Test of an Elbow Subjected to In plane Moment Loading -

As it is seen on the results in 1.4 and 2.4 the computed behavior differs greatly when the two codes are compared. This differences are probably due to the fact that the end effects are not taken in account in the simplified method.

3-3.4. Conclusion -

From these observations it can be concluded that the effect of the straight parts on elbows are important.
They are taken in account in the three dimensional shell analysis but not in the simplified method. Some improvements in the TEDEL program are now in progress but were not available in time for these computations.
Fig. 3.3.1 - Maximal radial displacement

90° Elbow

Déplacement radial maximal
le long d'une génératrice

COUDE à 90°
Fig. 3.3.2 - Maximal Radial Displacement 180° Elbow

Deplacement radial maximal le long d'une génératrice

COUDE à 180°
Fig. 3.3.3 - Horizontal displacement
180° Elbow

Deplacement $s$ en fonction du temps

COUDE à 180°
Fig. 3.3.4 - Rotation of the 180° Elbow

\[ \theta (\text{radiants}) \]

\[ \text{TEMPS (heures)} \]

COUDE à 180°
Fig. 3.3.5 - Rotation of a point 90° Elbow
(point number 12 of Tedel's mesh)
3.4. Influence of geometrical data -

3.4.1. Elevated Temperature Elastic - Plastic - Creep Test of an Elbow Subjected to In-Plane Moment Loading -

To see the influence of geometrical data a new calculation was done on the cited Benchmark Problem. The computer code used is the simplified method TEDEL. The material data are those given in reference 7.

The elbow is supposed to have a uniform thickness of 6.5 mm (termed nominal thickness in ref. 7).

The resulting deflection is given by figure 3.4.1. The computed deflection is enhanced by the added flexibility and these results are to be compared to those of 2.4 above.

Figures 3.4.2 and 3.4.3 gives the radial displacement and the strains obtained with this geometry. As it can be observed by comparing these results to the results of paragraph 2.4 geometrical data are of great importance for piping calculations and especially elbows.
Fig 3.4.1. Load deflection curve (elbow thickness = 6.5 mm, material data of Ref. 6)
3-5. Cost effectiveness -

All the computations are made on an IBM 360/91 computer. Table 3.5.1. gives a cost comparison between the simplified piping analysis computer code TEDEL and the three-dimensional shell analysis computer code TRICO. The memory is the central processor memory necessary to solve the problem it is given in k bytes. The total time is given but as it includes the compilation of the material data subroutine and linkage of the program two other times are given for comparison: that is, the stiffness matrix inversion time as well as the time to compute and assemble the stiffness matrix.

The differences in computing costs between a very efficient three-dimensional code and a simplified method is tremendous. This gives a very promising future for simplified methods. These methods are already more justified for the analysis of complete piping systems. After some improvements it will be possible to analyse accurately elbows, taking into account the change of cross-sectional shape and the effect of straight parts.
<table>
<thead>
<tr>
<th>Problem</th>
<th>Code</th>
<th>Memory (k bytes)</th>
<th>Total time (sec)</th>
<th>Cost (u.c.)</th>
<th>Stiffness matrix resolution time (sec)</th>
<th>Stiffness matrix computation time (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TRICO</td>
<td>600</td>
<td>770</td>
<td>1688</td>
<td>19.6</td>
<td>26.8</td>
</tr>
<tr>
<td></td>
<td>TEDEL</td>
<td>296</td>
<td>43</td>
<td>80</td>
<td>0.05</td>
<td>0.4</td>
</tr>
<tr>
<td>2</td>
<td>TRICO</td>
<td>600</td>
<td>560</td>
<td>1405</td>
<td>15</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td>TEDEL</td>
<td>300</td>
<td>36</td>
<td>78</td>
<td>0.6</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>TRICO</td>
<td>560 (1)</td>
<td>996</td>
<td>2568</td>
<td>14</td>
<td>16</td>
</tr>
<tr>
<td></td>
<td></td>
<td>560 (2)</td>
<td>1880</td>
<td>3700</td>
<td>17</td>
<td>17</td>
</tr>
<tr>
<td></td>
<td>TEDEL</td>
<td>260</td>
<td>323</td>
<td>591</td>
<td>0.7</td>
<td></td>
</tr>
</tbody>
</table>

**TABLE 3.5.1 - COST COMPARISON**

- **problem 1**: Elbow - Pipe Assembly Subjected to In-Plane Moment Loading at 593°C
- **problem 2**: Elevated - Temperature Elastic - Plastic - Creep Test of an Elbow
- **problem 3**: Room - Temperature Elastic - Plastic Response of a thin walled Elbow

- **TRICO**: three dimensional shell analysis program
- **TEDEL**: simplified piping analysis program
- **1 u.c.**: 0.5 $\text{U.S.}$ in July 1978 at the CEA computer center
- **(1)** first part loading history (cyclic loads)
- **(2)** second part loading history (monotonic loads)
3-6. Closure

The three piping Benchmark Problems proposed for solutions are analysed with two computer codes. One is based on a three dimensional shell finite element and the other on a simplified piping analysis method.

The three dimensional shell analysis computer code TRICO is presented. It is based on the global method for elastic-plastic-creep analysis. The global method is summarised as well as the \( \varepsilon^x \) - Hoffmann method for nonlinear structural analysis of beams and shells. The conjunction of these two methods produces an efficient program for the analysis of shells in three dimension. Accuracy is good and computer costs are kept very low. The main features of the global method is that it is based on limit load analysis and does not involve integration points in the shell thickness (When a more precise picture of plasticity through the thickness is necessary a multilayer shell element is used). The main feature of the \( \varepsilon^x \) - Hoffmann method is its internal iteration scheme for plastic strains computation which produces a stable and very efficient algorithm.

For the two piping Benchmark Problems involving creep the program TRICO gives results which do not agree with the measured ones. The differences are analysed in view of the sensitivity of plastic - creep analysis to the material data. It is asked if the chosen values for these two Benchmark Problems are applicable. This involves the necessity of some other Benchmark Problems including creep effects. For these futur Benchmark it is suggested to measure the material data on specimens taken on the structure or on an identical structure. Then it will be possible to see if the observed differences between computed and measured values are due more to the chosen constitutive equations or to the material data.
For the room-temperature Piping Benchmark Problem an option of the TRICO program is not available. It is not possible to combine large deflection and kinematic hardening. This produces poor agreement for the first part of the load history. (It is due to the changes of cross sectional shape of the elbow which are different in opening and closing of the elbow). The second part of the load history is computed with isotropic hardening (due to the monotonic loading) and the large deflection option. Good agreement is observed for a given deflection. The differences for a given load can be explained by the force measurement difficulties and by the fact that a very little change in load produces a large change in deflection.

The simplified piping method of the program TEDEL is presented in part two of this report. It is based on a beam type method and uses the already described features of the global method for the constitutive equations and of the $\epsilon^k$ - Hoffmann method for the numerical solution. A simple model for geometrical non-linearities is used.

The cost effectiveness of this method is very attractive and the results obtained are good enough for design purpose when it is necessary to compare the behavior of different structural shapes for a same component or structure. But the geometrical non-linearities model is too simple for very large deflection and the program is now in a modification stage for this problem. Another phenomena was pointed out : that is, the effects of straight pipes on elbows is not yet taken in account for the simplified model. The adequate modifications in the computer program are now in progress.

For the two 90 elbows in creep these end effects explain the differences in computed results between the TRICO and TEDEL programs. After modification of the TEDEL program it is hoped that these differences will be reduced.
For the 180° elbow at room temperature very good agreement is obtained between computed results and measured ones for the first part of the loading history. The loads here produce limited geometrical changes which are taken in account by the simple model. For the second part of the loading history very large displacements are encountered and the restraining effect of the straight parts becomes the most important phenomena.

Finally the six Benchmark Problems calculations and the supplementary calculations represented a very important work with much difficulties. But they provided the basis for some improvements in the computer programs. They also showed some weaknesses in the Benchmark Problems themselves.
REFERENCES

1. A. HOFFMANN, F. JEANPIERRE, M. LIVOLANT

2. A. HOFFMANN
   Programme TRICO. Analyse des structures tridimensionnelles composées de coques et de poutres. Note CEA - N 1930 - CEN Saclay.

3. A. HOFFMANN, T. CHARRAS
   Programme COCO. Maillage automatique de structures planes ou tridimensionnelles. Note CEA - N - 1937 CEN Saclay.

4. F. AURIOL et all.
   Fonctions ESPACE, TEMPS, VISU. Dépouillement automatique des résultats. Note CEA - N - 1928 CEN Saclay.

5. J.M. CORUM
   Appendix : Material property data for elastic - plastic - creep analyses of Benchmark Problems (in pressure vessels and piping verification and qualification of inelastic computer programs) ASME - G 0008.

6. W.I. GRIFFITH, E.C. RODABOUGH
   Tests at 593°C (1100°F) on a 101.6 mm (4 - in) sched - 10, Elbow-pipe Assembly subjected to in plane Moment Loading. Intern. Piping Benchmark Problems Nov. 1977.
7 A. IMAZU et all.
Elevated - Temperature Elastic - Plastic - Creep
Test of an Elbow Subjected to In plane Moment Loading.
Intern. Piping Benchmark Problems.
Nov. 1977.

8 D. BROUARD, R. ROCHE and B. VRILLON
Room - Temperature Elastic - plastic Response of a
thin walled elbow subjected to in plane bending loads.
Intern. Piping Benchmark Problems
Nov. 1977.

9 R. ROCHE, A. HOFFMANN
Global plastic models for computerised structural
analysis.

10 R. ROCHE
Modèle simple pour le calcul plastique d'une tuyauterie
Note CEA - N - 1872 CEN Saclay.

11 R. ROCHE, A. HOFFMANN
Inelastic piping flexibility analysis. A beam type
method
Paper II.3 Specialists' Meeting on high temperature
design technology of LMFBRs. Champion, Pa april 1976.

12 J. SPENCE and G. FINDLAY
Limit load for pipe bend under plane bending
2th Intern. Conf. on P.V.T.

13 A. HOFFMANN et all.
Programme TEDEL. Tuyauteries - Charpentes à 3
dimensions
Note CEA - N - 1930 CEN Saclay.
14 F. JEANPIERRE et all.
CEASEMT - System of finite element computer programs.
Use for inelastic analysis in liquid metal cooled reactor components.
IAEA/IWGFR
Specialists' meeting on high temperature structural design technology of LMFBRs - Champion, Pa 1976.

15 S.E. BOLT, W.L. GREENTREE
Experimental determination of plastic collapse loads for pipe elbows
ASME paper 71 PVP 37.

16 R. ROCHE, A. HOFFMANN, B. VRILLON
Piping systems, inelastic analysis - A simplified numerical method.
3th Intern. Conf. on P.V.T. Tokyo 1977.

17 D. BROUARD, R. ROCHE, B. VRILLON
Room - Temperature elastic - Plastic response of a thin walled elbow subjected to In-Plane bending loads.
Intern. Piping Benchmark Problems.
Nov. 1977.

18 W. PRAGER
The general theory of limit design

19 A. IMAZU et all.
Elevated temperature elastic - plastic - creep test of an elbow subjected to in-plane moment loading
Jour. of press. vess. tech.
May 1977

20 R.M. MELLO, D.S. GRIFFIN
Plastic collapse loads for pipe elbows using inelastic analysis
ASME paper 74 - PVP - 16
S.K. CHAN, Y. FAN, A. W. FILSTRUP, S.E. GABRIELSE
Verifying the plastic capabilities of a general purpose computer code. Pressure vessels and piping computer program evaluation and qualification
ASME G 00 124

G.E. FINDLAY, J. SPENCE
Stress analysis of smooth curved tubes with flanged end constraints.
University of strathclyde
February 1978.