MECHANICAL EVALUATION OF A NATURAL UF_6 TRANSPORT CONTAINER AT HIGH TEMPERATURE

K. Shirai (1), M. Wataru (1), A. Kosaki (1), T. Saegusa (1) and K. Shimamura (2)

(1) Central Research Institute of Electric Power Industry (CRIEPI)
1646, Abiko, Abiko-shi, Chiba-ken, 270-11, Japan
(2) Ishikawajima-Harima Heavy Industries Co., Ltd. (IHI)
1-15, Toyosu 3-Chome, Koto-ku, Tokyo, 135, Japan

SUMMARY

International Atomic Energy Agency (IAEA) revised the transport regulation for natural Uranium hexafluoride (UF_6) transportation taking into account chemical and radiological hazards in 1996. A supplementary fire test requirement (800°*C* for 30 minutes) was imposed on the natural UF_6 transport container. In 1996, Central Research Institute of Electric Power Industry (CRIEPI) and Nuclear Protection and Safety Institute (IPSN) terminated experimental joint research works with the aim to determine the thermal-physical behavior of UF_6 in a transport container under realistic fire conditions and to use the experimental data to validate a thermal-hydraulic numerical model. Now, they have started a new experimental joint research as to the rupture test of the 48Y-cylinder which will be terminated at the end of 1998. The purpose of this study is to evaluate numerically the mechanical integrity of this cylinder in the IAEA fire test conditions.

Firstly, pre-thermal-hydraulic numerical analysis of the 48Y-cylinder under the IAEA fire test condition was performed (Shirai, 1997). Nextly, the structural material model at high temperature for natural UF_6 transport container was proposed based on the CRIEPI's material tests (Kosaki, 1994) and applied to the ABAQUS computer code. According to the mechanical non-linear analysis results, it was found that it is necessary to evaluate the safety margin for the rupture of the 48Y-cylinder because considerable plastic and creep deformations are generated due to the temperature distribution of the cylinder and the inner pressure. This thermal-mechanical behavior of the container will be verified according to the rupture test results of the 48Y-cylinder until the end of 1998.

INTRODUCTION

UF_6, the raw material from which the fuel for nuclear power stations is obtained, is stored and transported in solid state in industrial containers called 48Y-cylinder. IAEA revised the transport regulation for natural UF_6 transportation taking into account chemical and radiological hazards in 1996. A supplementary fire test requirement (engulfing fuel fire of 800°C for half an hour, for a steel emissivity of 0.8 and flame emissivity of 0.9) was imposed on the natural UF_6 transport container. ASTM SA516 carbon steel for moderate and low temperature service is now used as the structural material for this type of cylinder. When the UF_6 transport container is involved in a fire, packaged UF_6 can easily be transformed from solid phase to liquid or gas phase at a comparatively low temperature, and can cause an inner pressure increase. The structural strength of the cylinder material also decreases with increasing temperature (Kosaki, 1994). Therefore, it is very important to evaluate the thermal-mechanical behavior of UF_6 cylinder under realistic fire conditions, especially the possibility of rupture of the cylinder.

DEFINITION OF THE PROBLEM

The UF_6 is a colorless solid at ambient temperature. The specific characteristics of the thermal behavior of UF_6 are its low temperature triple point (0.15MPa, 64°C), phase change and volume expansion. If the fire test recommended by the IAEA is imposed on the 48Y-cylinder, a very complicated heat transfer including boiling phenomena as shown in Fig.1 takes place and leads to a rapid increase in pressure in the last ten minutes of the fire test most likely up to the rupture of the cylinder. So, there are a lot of difficulties to get an accurate numerical calculation.

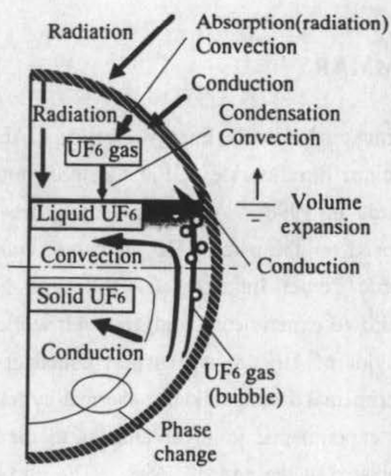


Fig.1 Diagram of Heat Transfer in the Cylinder

THERMAL-HYDRAULIC ANALYSIS

CRIEPI and IPSN performed the experimental joint research works (TENERIFE Program) to make clear the thermal-physical behavior of UF_6 in a transport container under realistic fire conditions which had been started in 1991 and terminated in March, 1996 (Sert et al., 1995). Its objective was to obtain experimental data and qualify the thermal-hydraulic computer codes. To evaluate and clarify the thermal-hydraulic phenomena of UF_6 in the container, an analysis and interpretation of physical phenomena have been carried out and a numerical model was elaborated by Commissariat A l'Energie Atomique (CEA) Grenoble (Pinton et al., 1995). A two-dimensional model using the finite element computer code called DIBONA was developed.

The specific physical phenomena as shown in Fig.1 considered in this model are expansion due to the density difference between solid and liquid UF_6 ,heat transfer during boiling, condensation of the vapor bubbles, equation of state of UF_6 and melting and sinking of solid UF_6. The thermal-hydraulic numerical analysis with the DIBONA computer code (Shirai et al., 1997) was performed to define the temperature distribution of the container and the inner pressure caused by a fire described in the IAEA regulation. Fig.2 shows the numerical results for the maximum container temperature and the inner pressure for IAEA fire test conditions.

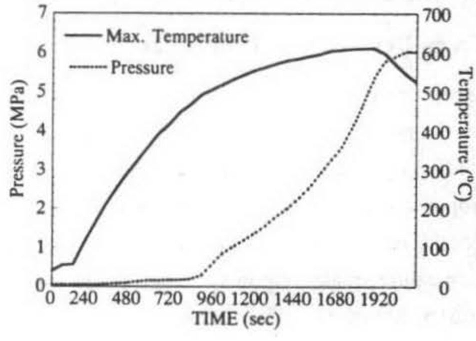


Fig.2 Thermal-Hydraulic Numerical Analysis Results

MATERIAL TEST AT HIGH TEMPERATURE

ASME SA516 carbon steel for moderate and lower temperature service is currently used as the structural material for natural UF_6 container. According to the thermal-hydraulic analysis results, a considerable inner gas pressure rise is generated. Simultaneously, the structural strength of the container decreases with increasing temperature, so the possibility of rupture of the packaging has been studied (Kosaki et al. 1994). As SA516 steel is not generally used for high temperature work, material characteristics had not been available above 500°*C*.

High Temperature Tensile Test

Tensile tests from room temperature up to 900°*C* were conducted using several base metal and seam-welded joints of SA516 produced in USA, France and Japan (A. Kosaki, 1994). The strain rate of the test was $5 \times 10^{-3}\ min^{-1}$ and over 0.2% proof stress was $6 \times 10^{-2}\ min^{-1}$. Fig.3 shows an example of the test results. The tensile strength values and 0.2% proof stress decrease with increasing temperature. No influence of phase transformation was observed and all materials whether produced in USA, France or Japan

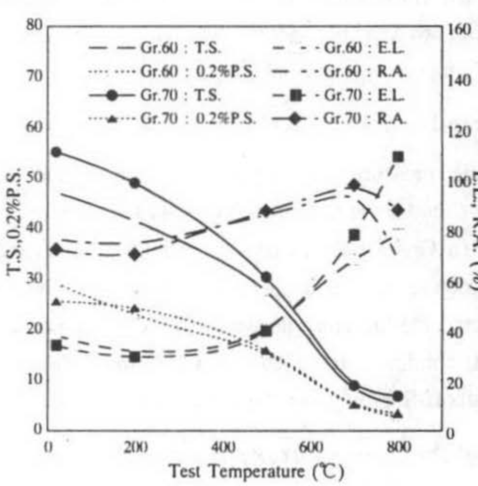


Fig.3 Tensile Properties of SA516 Steels
(French Materials : Base metal)

show the same tensile tendency. On the other hand, the values of the reduction area and elongation increase with increasing temperature. The position of rupture of seam-welded joints was the base metal at temperatures from room temperature up to 800°C, but at 900°C, the rupture position was the weld metal.

Creep Deformation Properties

Short-time uni-axial creep tests and interior pressure creep tests using cylindrical test pieces were conducted at 600-900°C and at various stress levels at each temperature (stress range : 8-45MPa) by using SA516 Gr.65 base metal, and the creep constitutive equation was originally proposed especially paying attention to the high temperature service region beyond 700°C (A. Kosaki et al., 1994). To extrapolate the creep deformation in moderate and lower temperature service region with high stress condition, modified creep deformation formula for SA516 base metal, which describe primary and secondary creep, are proposed as follows.

$$\varepsilon_C = \varepsilon_T + \dot{\varepsilon}_S\, t \qquad (1)$$

$$\varepsilon_T = \varepsilon_T^S\left[1 - exp\left\{-1.723\left(\dot{\varepsilon}_S\, t\right)^{0.538}\right\}\right] \qquad (2)$$

$$\varepsilon_T^S = exp(0.0592\bar{\sigma})\, exp\left(-\frac{5060}{T + 273} + 1.21\right) \qquad (3)$$

$$\dot{\varepsilon}_S = 1.670 \times 10^{10}\, exp(0.639\bar{\sigma})\, exp\left(-\frac{4.790 \times 10^2\bar{\sigma} + 2.748 \times 10^4}{T + 273}\right), \quad T < 723°C$$

$$\dot{\varepsilon}_S = 5.189 \times 10^{9}\, exp(-0.631\bar{\sigma})\, exp\left(\frac{2.2 \times 10^3\bar{\sigma} + 2.663 \times 10^4}{T + 273}\right), \quad 723°C < T < 845°C \qquad (4)$$

where, ε_c = creep strain (%), ε_T = transition creep strain (%), ε_T^S = saturated transition creep strain (%), $\dot{\varepsilon}_S$ = minimum creep strain rate (%/hour), t = time(hour), $\bar{\sigma}$ = Mises stress (MPa), T = test temperature (°C)

Fig.4 shows the relationship between the test temperature and calculated minimum creep strain rate with the experimental values.

Creep Rupture Criteria

Interior pressure creep rupture tests using cylindrical test pieces were also conducted at 600-800°C and at various stress levels at each temperature (Mises stress range : 30-140MPa) using SA516 Gr.65 base metal, and the life-time formula was also proposed especially paying attention to the high temperature service region beyond 700°C (M. Wataru et al., 1995). To estimate the life-time adequately in moderate and lower temperature service region with high stress condition, modified life-time formulae for SA516 base metal is proposed based on the Goldhoff-Sherby parameter method as follows.

$$log(t_r) = \left(\frac{1}{T + 273} - 0.00125\right)\cdot(275\bar{\sigma} - 1336) + 3 \qquad (5)$$

where, t_r = rupture time (h)

Fig.5 shows the relationship between the Mises stress and rupture time.

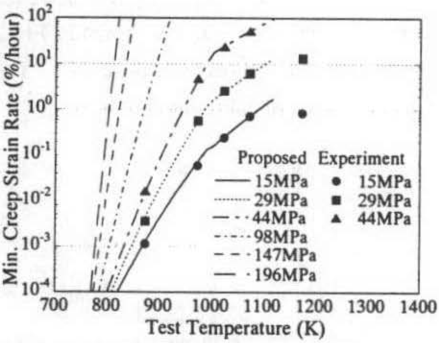


Fig.4 Relationship between Test Temperature and Minimum Creep Strain Rate

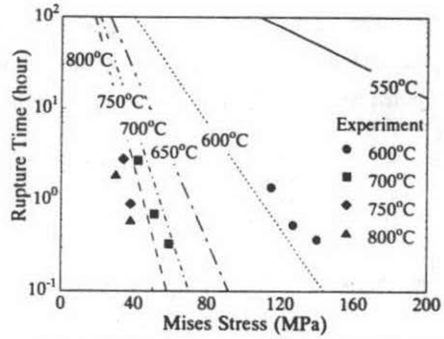


Fig.5 Relationship between Mises Stress and Rupture Time

THERMAL-MECHANICAL ANALYSIS

Plastic Deformation

Firstly, the thermal-mechanical non-linear analysis was performed with ABAQUS computer code. A generalized plane strain quasi-three-dimensional model was applied in the analysis. Fig.6 shows the finite element model. Two eight-nodes solid elements are placed through the thickness so that bending effects will be taken into account properly. Considering the uniformity of the axial deformation, it is assumed that the cross section of the model must constrain the plane strain condition during deformation.

%	θ (deg.)
60	9.08
65	13.8
70	18.6
75	23.8
80	29.5
85	35.8
90	43.4

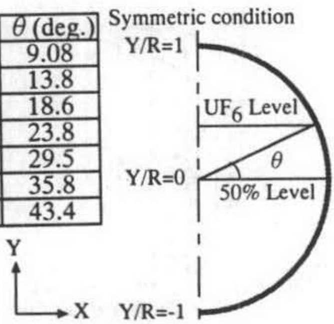
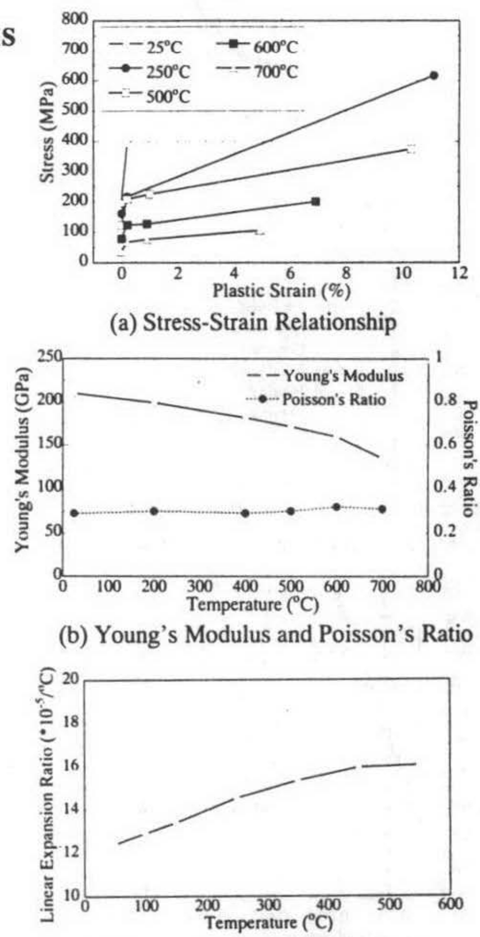


Fig.6 Finite Element Model



(a) Stress-Strain Relationship

(b) Young's Modulus and Poisson's Ratio

(c) Thermal Linear Expansion Ratio

Fig.7 Mechanical Properties used for Calculation

The thermal loading condition is based on values obtained in calculations using DIBONA-2D code. The loading consists of temperature distribution and inner pressure as shown in Fig.2 applied on the edge of the elements located inside the cylinder and atmospheric pressure. The additional heat input from the stiffening ring of the cylinder was not taken into account.

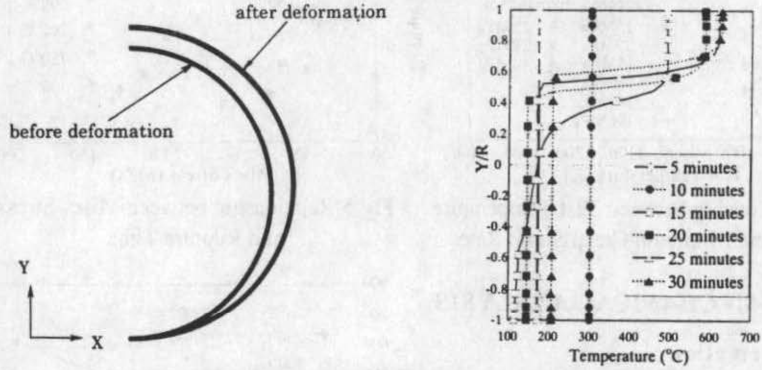


Fig.8 Deformation after 30 minutes and Thermal Distribution

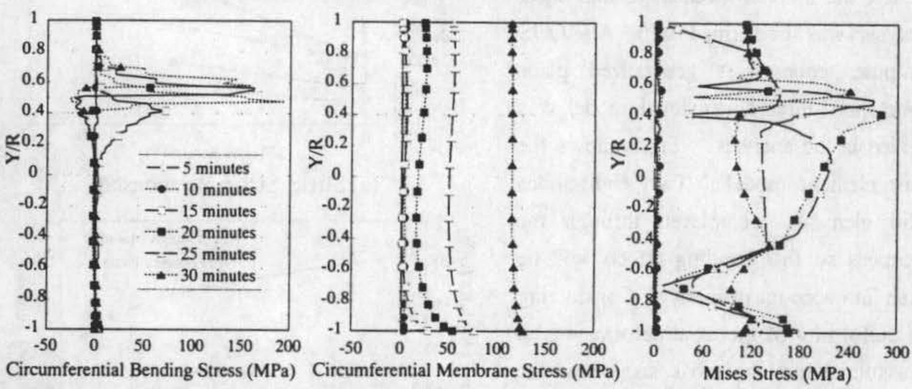


Fig.9 Distributions of Circumferential Bending and Membrane stresses and Mises stress

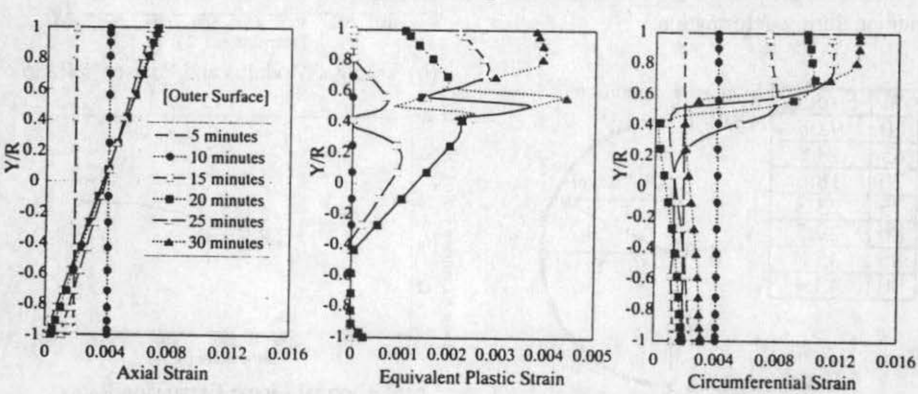


Fig.10 Distributions of Axial, Circumferential and Equivalent Plastic Strain

Referring the material tests as shown in Fig.3, the mechanical properties of the SA516 Gr.65 ,such as stress-strain relationship, are used as shown in Fig.7. As a constitutional law, elasto-plastic model (multilinear isotropic hardening model) was applied, and Von-Mises yield criterion and Prandtl-Reuss flow rule were used. Fig.8 shows the deformation after 30 minutes thermal loading and the temperature distribution. Fig.9 shows the distribution of the circumferential bending and membrane stresses and Mises stress. Maximum circumferential membrane stress value was 120MPa, which was generated by the inner pressure. Mises stress had a maximum value just below the UF_6 liquid level and decreased considerably at the UF_6 level. Fig.10 shows the distributions of the axial, circumferential and equivalent plastic strain. It is found that the maximum plastic strain was reached to only 0.5% .

Creep Effect

Secondly, creep effect was investigated using proposed creep deformation formulae because high stress region at intermediate temperature (about 600°C) was observed in the thermal-mechanical analysis results. Fig.11 shows the creep strain distribution. It is found that creep strain increased after 25minutes and exceeded 10%. To evaluate the possibility of rupture of the 48Y-cylinder, according to the modified life-time formula represented by equation (5), the rupture time was estimated by Robinson's law method. In this method, the rupture is assumed to occur when the sum of the creep damage factor exceeds the threshold value f_s as follows.

$$D = f_s \cdot \sum \left(t_i / t_{ri} \right) \tag{6}$$

where, D = creep damage factor, t_i = time at certain constant condition, t_{ri} = rupture time calculated by equation (5), f_s = safety factor

For the conservative estimation of rupture time, the safety factor of creep damage f_s should be smaller than 0.6 (M. Wataru et al., 1995). Fig.12 shows the distribution of the creep damage factor. It is found that creep damage factor reached to 0.56 and there would be a possibility of rupture of 48Y-cylinder in a full engulfing fire.

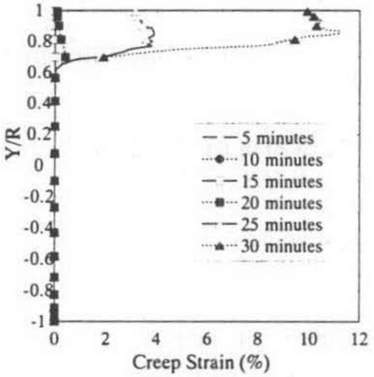


Fig.11 Creep Strain Distribution

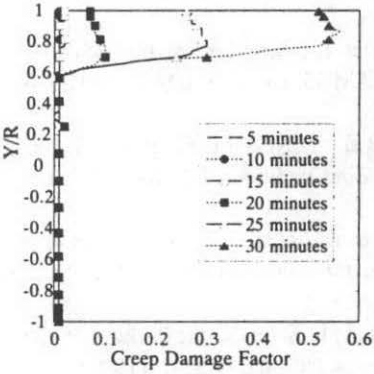


Fig.12 Creep Damage Factor Distribution

CONCLUSION

A supplementary fire test requirement (800°*C* for 30 minutes) was imposed on the natural UF_6 transport container. To evaluate the mechanical integrity of this cylinder in a fire, especially the possibility of rupture, material tests and thermal-mechanical analysis with ABAQUS code were performed. The outline of the contents and results are summarized below.

(1) Pre-thermal-hydraulic numerical analysis of the 48Y-cylinder under the IAEA fire test was performed with DIBONA-2D code developed by CEA Grenoble. The temperature distribution of the cylinder and the inner pressure due to the UF_6 gas were estimated. It is found that the upper part of the cylinder was heated up to 620°C and the inner pressure continued rising to a pressure near 6MPa.

(2) For the steel SA516 Gr.65, uni-axial tensile and creep tests and interior pressure creep rupture tests were performed to get material data for numerical calculations at high temperature. A thermal-mechanical numerical analysis for a 48Y-cylinder was performed with ABAQUS code. The maximum equivalent plastic strain reached to only 0.5%.

(3) As high stress region at intermediate temperature (about 600°C) was observd in the analysis results, creep effect was investigated using proposed creep deformation and damage factor formulae. It was found that it is necessary to evaluate the safety margin for the rupture of the 48Y-cylinder because considerable creep deformation will be generated by the temperature distribution and the high inner pressure. The thermal-mechanical characteristics of the cylinder under the IAEA fire test requirement will be verified according to the rupture test results of the 48Y-cylinder until the end of 1998.

REFERENCES

Kosaki et al., Material properties of a natural UF_6 Transport Cylinder Vessel at High Temperature, Int. J. of Radioactive Materials Transport, Vol.5, Nos2-4., (1994).

Pinton et al., Modeling of the Behavior of a UF_6 Container in a Fire -DIBONA Codes, PATRAM'95, Las Vegas, Nevada, U.S.A., (1995).

Sert. et al., TENERIFE Program : Thermalphysical Behavior of UF_6 in a Transport Container Under FireConditions, PATRAM' 95, Las Vegas, Nevada,U.S.A., (1995).

Shirai et al., Mechanical Evaluation of a Natural UF_6 Transport Cylinder Vessel at High Temperature, SMiRT14, Lyon, FRANCE, (1997).

Wataru et al., Safety of the Package of Natural UF_6 Under Fire Conditions, PATRAM'95, Las Vegas, Nevada, U.S.A., (1995).