Example Evaluation of a Representative Heat Pipe Test Article Design for Structural Acceptability using ASME Design Rules

Applied Materials Division

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free via DOE's SciTech Connect (<http://www.osti.gov/scitech/>)

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):

U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

Example Evaluation of a Representative Heat Pipe Test Article Design for Structural Acceptability using ASME Design Rules

Applied Materials Division
Argonne National Laboratory

September 2020

Prepared by

Guosheng Ye
Mark C. Messner
T.-L. Sham

ABSTRACT

This report describes a design analysis of a test article mimicking a core block of a heat-pipe microreactor using the ASME Boiler & Pressure Vessel Code Section III, Division 5, Subsection HB, Subpart B rules. These rules cover the design and construction of elevated temperature nuclear reactor structural components, like the core block of a heat-pipe microreactor emulated by the test article. The purposes of the report are to: (1) provide a step-by-step example of applying the ASME rules to a reasonable core block geometry and corresponding loading conditions, (2) to provide design feedback on the test article, and (3) to evaluate the ASME rules for use in the design of complex heat-pipe microreactor components. The test article design has a relatively short ASME design life, limited by high thermal stresses. The current ASME rules are adequate for evaluating the core block component designs, though they could be optimized to increase the design efficiency and reduce the amount of designer/analyst time required to execute the design process. The report covers both the current, 2019 edition ASME rules and modified, currently unapproved, rules designed to improve the design process for components with complex geometries and high secondary stresses, like the core block test article.

TABLE OF CONTENTS

Abstract	i
Table of Contents	iii
List of Figures	v
List of Tables	vii
1 Introduction	1
2 Analysis Report	3
2.1 Objectives of the analysis	3
2.2 Background	3
2.3 Scope of the analysis	5
2.4 Acceptance criteria	5
2.5 Methodology	8
2.6 Assumptions	13
2.7 Analysis Inputs	14
2.8 Calculation Results	19
2.9 Discussion	24
2.10 Analysis summary	26
3 Conclusion	29
Acknowledgements	31
References	33

LIST OF FIGURES

Figure 1. Schematic of power generation by heat-pipe microreactors [1].	3
Figure 2. Heat-pipe microreactor geometry in FEA.	5
Figure 3. Flow diagram for elevated temperature analysis.	6
Figure 4. Assumed flow transient applied in the analysis.	14
Figure 5. Meshes in the heat-pipe microreactor FEA model. (a) core block, (b) heat exchanger, hot section, (c) heat exchanger, cold section. (d) heat pipes in the heat exchanger hot section, (e) heat pipes in the heat exchanger cold section.	15
Figure 6. Mechanical loads and boundary conditions. (a) constraints, (b) pressure load.	16
Figure 7. Thermal loads/boundary conditions in the core block.	17
Figure 8. Radiative heat transfer surfaces.	18
Figure 9. Convective heat transfer boundary conditions. (a) at heat pipe, (b) coordinate system used to evaluate h and $T0$, (c) at hot tube bank, (d) at cold tube bank.	18
Figure 10. (a) thermal insulation inside wall, (b) constant temperature at outside wall.	19
Figure 11. Stress linearization locations at different parts of the heat-pipe microreactor. (a) core block, (b) heat pipe, (c) heat exchanger-hot, (d) heat exchanger-cold.	20
Figure 12. Contour plot of equivalent plastic strain for design limit check with EPP method.	21
Figure 13. Equivalent plastic strain contour plot for service limit check.	22
Figure 14. Contour plot of damage parameter D at 100,000 hours service life for local check.	23
Figure 15. Locations for creep-fatigue damage check.	23
Figure 16. Total stress relaxation and comparison with pressure stress.	25

LIST OF TABLES

Table 1. Pseudo yield strength values for EPP analysis.	16
Table 2. Stress analysis results at different locations for design limit check.	21
Table 3. Stress analysis results at different locations for service limit check.....	22
Table 4. Strain and deformation limit check.....	23
Table 5. Fatigue damage calculation results.	24
Table 6. Creep damage calculation results.	24
Table 7. Creep-Fatigue damage calculation summary.....	24
Table 8. Service limit check with thermal stress as primary load.	26

1 Introduction

This report evaluates a heat-pipe microreactor test article design from Los Alamos National Laboratory (LANL) against the ASME Boiler & Pressure Vessel Code Section III, Division 5, Subsection HB, Subpart B rules covering the design and construction of elevated temperature, Class A nuclear reactor components. The LANL test article aims to evaluate heat transfer in future heat-pipe microreactor designs – electrical heating replaces nuclear heating in the test article. While primarily aimed at testing heat transfer, the test article is a reasonable test bed for evaluating the component design against design criteria for high temperature reactor structural components.

This report then has several purposes:

1. To provide a detailed example of the application of the ASME Section III, Division 5 rules to a realistic heat-pipe microreactor geometric design with realistic steady-state thermal loads.
2. To provide an evaluation of this particular core-block and heat exchanger design, representative of a heat-pipe microreactor, against the ASME rules.
3. As a sample problem for evaluating the design rules themselves for use in future heat-pipe microreactors.

The ASME Section III, Division 5 design rules were developed with traditional advanced reactor designs in mind, e.g., sodium fast reactors and gas-cooled high temperature reactors. However, the analysis here demonstrates that they are applicable to the core block and heat exchanger of a heat-pipe microreactor within the ASME Section III scope. The analysis report that forms the bulk of this document provides a step-by-step application of the ASME rules to the test article design. The conclusion is that, viewed as a representative heat-pipe microreactor, the test article would have a very short ASME design life, largely dictated by high thermal stresses inducing creep-fatigue damage in critical locations of the core block and heat exchanger. The thermohydraulics of the reactor or, to a lesser extent, the geometric design of the core block and heat exchanger, could be altered to extend the design life of the reactor system as a whole.

The ASME rules are not optimized for use in components where the thermal stresses control the design nor did the original Code rules envision complicated 3D geometries like core blocks. A main component of the ASME approach is the allowable stress criteria based on the component primary load. In the core block test article the primary load is essentially negligible and so the current primary load rules are not helpful in determining the component geometric design. Therefore, in addition, to the current rules in the 2019 edition of the ASME Code, this report applies a new draft, unapproved set of primary load design rules. These draft rules were designed to include the effect of high thermal stresses earlier in the ASME design process and to be broadly applicable to all component types, including complicated 3D geometries. The results show these new design rules fall somewhere in between the current Division 5 primary load design rules, which produce a long design life, and the creep-fatigue rules, which produce a very short design life. As such, they might be a reasonable basis for the initial, conceptual design of heat-pipe microreactor components.

Several assumptions had to be made in executing the design analysis. The most important was that the actual test article is made from 316L stainless steel. This material is not approved for

Class A construction and, in any event, there is insufficient test data on 316L to produce accurate design data. The report assumes the component material properties are those for wrought 316H stainless steel, which is an approved Class A material. Wrought 316L is weaker than wrought 316H at elevated temperatures and so the analysis conducted in this report is not a conservative analysis, assuming that 316L follows a similar trend. But the trend provided by the analysis results are instructional.

Additionally, fully executing the ASME design rules requires the definition of loading transients – how the component will be loaded, held at steady state, and unloaded. For the test article a transient thermohydraulic analysis could supply the structural loading conditions. However, this analysis was not available at the time of this report. Instead, we assume a simple, slow standard operating transient based on the steady state conditions provided by LANL.

LANL provided the component geometry and steady-state loading conditions in the form of an ANSYS analysis deck. ANL took this initial model, increased the mesh resolution to produce more accurate stresses, and modified the steady-state loading conditions into a reasonable plant transient cycle. A finite element analysis (FEA) of this model provides the stress analysis results used in the ASME design.

The report has only two main sections. Chapter 2 is a complete analysis report of the component, starting from the LANL model and the assumptions described previously. This report walks through the step-by-step analysis of the component, provides the permissible design lives for the various ASME design criteria and the current and draft design rules, and comments on the component geometric design and the application of the ASME rules. Chapter 3 is then a commentary and summary of the design report, focusing on the features of the LANL design resulting in high thermal stresses and a correspondingly limited design life and on potential improvements to the ASME rules to optimize the design procedure for complicated heat-pipe microreactor components, like the core block and heat exchanger, and for structures with high thermal stresses relative to the component primary stress.

2 Analysis Report

2.1 Objectives of the analysis

A heat-pipe microreactor was designed to harness the heat produced during nuclear reaction to generate electricity (~5MW). Its size is much smaller than the size of traditional advanced reactors (e.g., sodium fast reactors and high temperature gas-cooled reactors). As with design evolving processes for other advanced reactor systems, there are technical challenges^[1] for the heat-pipe microreactor. They are possible damage due to thermal stresses introduced by thermal gradients at high temperatures and fatigue damage due to large stress variation during heat-pipe microreactor heat up and cool down operations. Estimation of the damage is critical for safe operation of the design. This report shows our efforts to apply the construction rules of the ASME BPVC Section III, Division 5^[2] to the heat-pipe microreactor design for damage estimation and allowable service life prediction. Also, as the current code was developed for the design of larger advanced reactors, such as sodium-cooled fast reactors and high temperature gas-cooled reactors, recommendations for code modification are given for code improvement based on engineering principles.

This specific analysis covers a test article geometry that is representative of a heat-pipe microreactor design. Electrical heating replaces the nuclear heating in this test article. The design itself is configured to test the heat-pipe microreactor heat transfer systems, but it can also be used as a test bed to gauge the structural life of a similar heat-pipe microreactor design.

2.2 Background

In places where power grids are hard to reach, or locations just hit by natural disasters, a transportable, robust power source is needed. A heat-pipe microreactor could fulfill such a need by using the heat generated in the reactor to produce electricity. Since it utilizes heat pipes for heat transfer, there is no fans or pumps in the system, thus it is much simpler to operate and easier to control. A schematic of a heat-pipe microreactor is shown in Figure 1, within which the core block is the location where nuclear reaction occurs and gives off heat. The heat pipes then take the heat from the core block and bring it to the heat exchanger, where the heat is transferred to the secondary and an open-air Brayton cycle is employed for power generation.

Figure 1. Schematic of power generation by heat-pipe microreactors [1].

As mentioned previously, there are some technical challenges in heat-pipe microreactor design, two prominent ones are the high temperature damage and fatigue damage. Thermohydraulic analysis results show that the core block of the heat-pipe microreactor operates at a pressure of 20 barg, which is low compared to 100 Barg or higher pressure in most steam power plant operations. However the temperature in heat-pipe microreactor can be as high as 700 °C and therefore high temperature creep damage is inevitable, especially at those webs between fuel rod holes, which are as thin as 1.75mm^[1]. On the other hand, due to low thermal conductivity of SS316 thermal stresses will be high during temperature ramping up and cool down which may lead to large stress variations and non-negligible fatigue damage. High temperature creep damage can result in component rupture, while fatigue damage can cause crack initialization, growth and eventually component failure. If both creep and fatigue occur at high temperature, either sequentially or simultaneously, creep-fatigue interaction will accelerate the damaging process. ASME Boiler and Pressure Vessel Code Section III, Division 5^[2] was developed to provide construction rules to guard against structural failure modes, with reasonable assurance for the integrity of the components of high temperature reactors. The failure modes addressed by Section III, Division 5, Subsection HB, Subpart B are:

- Plastic collapse and creep rupture
- Excessive deformation due to creep and/or ratcheting
- Crack initialization due to creep-fatigue interaction

This report walks through the test article design evaluation process based on the ASME Section III, Division 5, Subsection HB, Subpart B rules. It also provides recommendations for future code amendments, because the current code is based on design experience of traditional ad-

vanced reactors, e.g., high temperature gas-cooled reactors and sodium fast reactors with cylindrical geometric shapes, and their operation modes are different from those for heat-pipe micro-reactors.

2.3 Scope of the analysis

The scope of this analysis was limited to the core block test article with electrical heating holes and heat pipe holes, plus heat pipe and the heat exchanger. The design temperature is 720 °C and design pressure is 20 barg. The geometry of the test article in the FEA is displayed in Figure 2.

Figure 2. Heat-pipe microreactor geometry in FEA.

2.4 Acceptance criteria

The path to check for the design acceptance is shown in Figure 3. Four different checks were conducted for the test article design which include design limit check, Level A Service limit check, strain and deformation limit check and creep-fatigue damage check. The acceptance criteria for each check are described in Figure 3, and Section 2.4 of the report provides more detail

of that figure, such as how to obtain P_m , P_L , P_b and Q_R through stress linearization in the design by elastic analysis method, creep-fatigue damage calculation procedure, etc.

To reduce conservatism introduced by elastic method, elastic-perfectly plastic (EPP) methods, recommended by a Code Case draft^[3], were also applied as part of the efforts on ASME code improvement.

Figure 3. Flow diagram for elevated temperature analysis.

2.4.1 Design limit check

Design limit check considers primary membrane P_m (or P_L the local primary membrane stress) and primary bending stress P_b , and material property S_o is based on extrapolated 100,000 hour

material property data, Equation (1) and Equation (2) must be satisfied to pass the design limit check.

$$P_m < S_0 \quad (1)$$

$$P_L + P_b < 1.5S_0 \quad (2)$$

2.4.2 Level A service limit check

Level A service limit is for checking component routine loads, and Equations (3), (4) and (5) must be fulfilled to pass the check. In these equations, K , K_t are parameters related to component wall geometry, S_m is time-independent allowable stress, S_t is time-dependent allowable stress and S_{mt} is the lower of S_t and S_m at specific time and temperature. The reason for service limit check is that some service load is short-term and should not be considered in design check, otherwise the results would be too conservative^[2], on the other hand, short-term load should be checked for component's safe operation.

$$P_m < S_{mt} \quad (3)$$

$$P_L + P_b < 1.5S_m \quad (4)$$

$$P_L + P_b/k_t < S_t \quad (5)$$

For Equation (3), the use-fraction sum has to be checked to ensure

$$\sum_i \frac{t_i}{t_{im}} < B \quad (6)$$

Where t_i is the total duration at P_{mi} at temperature T_i ; t_{im} is the maximum allowed time for load stress intensity P_{im} at temperature T_i , determined from S_t -vs-time plot in [2]; $B \ll 1$, the exact value is based on design specification. For Equation (5), the use-fractions have also to be checked, to meet the requirement of (7).

$$\sum_i \frac{t_i}{t_{ib}} < 1 \quad (7)$$

Where t_i is the total duration at $P_L + P_b/k_t$ at temperature T_i , t_{ib} is the maximum allowed time obtained from S_t -vs-time Table HBB-I-14.4B in [2], at load equal to $P_L + P_b/k_t$ and temperature T_i .

2.4.3 Strain and deformation limit check

In order for the component to function well, its deformation has to be bounded during service, i.e., averaged strain through wall thickness $< 1\%$, strain at the surface $< 2\%$, and strain at any location $< 5\%$. Test No. B-2 in Appendix HBB-T of [2] was chosen for the strain and deformation limit check, and a creep stress parameter Z was obtained through P_L , P_B , Q_R , (the secondary stress range) those stress linearization results and a material property value S_{YL} . Section 5.4 will give more detail on the calculation procedure. The total strain during component service life ϵ_{total} can be obtained from HBB-T-1800 plots in [2], for stress at the level of $1.25 \cdot Z \cdot S_{YL}$ and operation temperature T , and it should satisfy Equation (8) to pass the check.

$$\epsilon_{total} < 1\% \quad \text{for base metal} \quad (8)$$

2.4.4 Creep-Fatigue damage check

On the damage due to creep and fatigue, for a design to be acceptable it should satisfy^[2]

$$\sum_{j=1}^p \left(\frac{n}{N_d} \right)_j + \sum_{k=1}^q \left(\frac{\Delta t}{T_d} \right)_k \leq D \quad (9)$$

Where $(n)_j$ is the number of type j cycle; $(N_d)_j$ is the number of allowable type j cycle, determined from Figure HBB-T-1420-1B of [2], the design fatigue range for SS316, corresponding to the maximum metal temperature occurring during the cycle; p is the number of cycle types, q is the number of time intervals at specified temperatures during whole service life; $(\Delta t)_k$ is the duration of the time interval k , $(T_d)_k$ is the allowable time duration for the time interval k at a given stress and the maximum temperature; D is the total allowable creep-fatigue damage, which can be obtained from Figure HBB-T-1420-2 of [2]. The procedure for calculating $(N_d)_j$ and $(T_d)_k$ are given in Section 5.5, and the contents are from Paragraph HBB-T-1430 of [2].

2.5 Methodology

2.5.1 Elastic method

Before elastic analysis results, i.e., P_m , P_L , $P_L + P_b$ and Q_R , described in Part 4 of the report, are compared with code criteria, stress components have to be linearized to obtain those values. Based on HBB-3200 of [2], P_m , P_L , P_b are stress intensities due to primary load and Q_R is stress intensity due to secondary load. To calculate P_m , first obtain the stress components $\sigma_{11,m}$, $\sigma_{22,m}$, $\sigma_{33,m}$, $\sigma_{12,m}$, $\sigma_{23,m}$ and $\sigma_{13,m}$ according to (10), then calculate the three principal stresses $\sigma_{1,m}$, $\sigma_{2,m}$ and $\sigma_{3,m}$, and P_m is obtained through (11).

$$\sigma_{ij,m} = \frac{1}{t} \int_0^t \sigma_{ij} dx \quad i,j = 1,2,3, t \text{ is the wall thickness} \quad (10)$$

$$P_m = \max(\sigma_{1,m}, \sigma_{2,m}, \sigma_{3,m}) - \min(\sigma_{1,m}, \sigma_{2,m}, \sigma_{3,m}) \quad (11)$$

P_L calculation follows the same procedure, except the membrane stress obtained is the local one.

P_b is the primary bending stress. Its calculation procedure is similar to that for P_m , except $\sigma_{ij,b}$, the bending part of the stress component is calculated as

$$\sigma_{ij,b} = \frac{6}{t^2} \int_0^t \sigma_{ij} \left(\frac{t}{2} - x \right) dx \quad (12)$$

According to [2], bending stresses are calculated only for the local hoop and meridional stress components, and not for the those parallel to the stress classification line (SCL) or in-plane

shear stress. In other words, if the stress classification line (SCL) is along x axis of the coordinate system, $\sigma_{11,b}$, $\sigma_{31,b}$, and $\sigma_{12,b}$ should be set to 0 before calculating P_b .

The procedure for Q_R calculation is similar to that for P_m too, except the stress components are due to secondary loads and include both membrane and bending stresses.

2.5.2 Elastic-Perfectly Plastic method

To do elastic analysis, stress linearization is inevitable, and the stress linearization location and categorization have been sources of arguments on the effectiveness of the analysis for years. With elastic-perfectly plastic (EPP) methodology proposed in a Code Case draft^[3], stress linearization and categorization are unnecessary, and through limit-load like FEA procedure, design limit check and service limit check can be accomplished efficiently.

2.5.2.1 Design limit check

A pseudo yield stress, which is equal to $0.87S_0$, is applied in a FEA model with a linear elastic-perfectly plastic material model. During analysis the loads ramp up from 0 values, when it reaches targeted design loads and the solution still converges, then the design passes the design limit check.

2.5.2.2 Service limit check

The procedure is similar to that for design limit check, except the pseudo yield strength is equal to $0.87S_{mt}$ related to target service life and temperature and the loads ramp up towards the targeted service load.

2.5.3 Local check

The aforementioned EPP method is mainly for ductile materials. For brittle material, the deformation is more localized towards its failure which requires an additional local check. Local check apply linear elastic material model with creep constitutive equations^[3] as follows,

$$\dot{\epsilon}_c = A \left(\sinh \frac{\beta \sigma_v}{n} \right)^n \exp \frac{-Q}{R(T+273.15)} \quad (13)$$

with constants

$$A = \begin{cases} 5.6229 \times 10^{12} & 427 \leq T < 538 \\ -7.8535 \times 10^{15} + 9.6933 \times 10^{12}(T + 273.15) & 538 \leq T < 579 \\ 5.28787 \times 10^{-6} \exp \left(\frac{39057.1}{T + 273.15} \right) & 579 \leq T < 649 \\ 6.03371 \times 10^{10} \exp \left(\frac{4967.76}{T + 273.15} \right) & 649 \leq T \leq 816 \end{cases}$$

$$\beta = 145.037681[-4.257 \times 10^{-4} + 7.733 \times 10^{-7}(T + 273.15)]$$

$$n = \begin{cases} 4.6 & 427 \leq T < 538 \\ -80.9236 + 0.105455(T + 273.15) & 538 \leq T < 579 \\ 50.1336 - 0.0482143(T + 273.15) & 579 \leq T < 649 \\ 14.4647 - 9.54954 \times 10^{-3}(T + 273.15) & 649 \leq T \leq 816 \end{cases}$$

$$Q = 67000$$

$$R = 1.987$$

Where $\dot{\epsilon}_c$ is the creep strain rate, T is the temperature in degree Celsius, σ_v is Von Mises' stress, and $D_c^i(x)$, the accumulated creep damage for a time duration t_i , can be expressed as

$$D_c^i(x) = \int_0^{t_i} \frac{dt}{t_M(\sigma_e(x,t))} \quad (14)$$

Where t_M is the minimum time to rupture at $\sigma_e(x, t)$, which is the effective stress at each location x at time t , and it is defined as

$$\sigma_e(x, t) = \alpha \sigma_1(x, t) + (1 - \alpha) \sigma_v(x, t) \quad (15)$$

In Equation (15), σ_1 is the maximum principal stress and α is equal to 0.5 based on the information in [3]. For FEA, a user defined subroutine for creep was applied and accumulated creep damage was regarded as a state variable, which can be shown on a contour plot, and $D_c \geq 1$ indicates the material at that location has been damaged.

2.5.4 Strain and deformation limit check procedure

The ASME BPVC Section III, Division 5, Paragraph HBB-T-1320 and Paragraph HBB-T-1330 of Appendix HBB-T provide different options for strain and deformation limit check procedures. Among them Tests No. A-1, No. A-2 and No. A-3 do not require the parameter Z value (these three tests are all "Pass" or "No Pass" type checks), which is necessary for the following creep-fatigue damage calculations, Test No. B1 requires negligible peak stress (this is not true in our case, and is shown in stress linearization results at some locations of the heat-pipe microreactor, where the peak stresses are 300% or higher than membrane + bending stress due to temperature gradients cross the wall) and Test No. B3 can only be applied to axisymmetric structure with axisymmetric loads, and at locations away from discontinuities (not applicable to heat-pipe microreactor geometries, which are not conventional thin wall tubes, i.e., biaxially-loaded vessel, upon which Bree diagram was obtained^[4], instead, core block of the heat-pipe microreactor is a structure with different sizes of holes and heat exchanger is a thick wall hexagon container), regarding all the above factors, Testing No. B2 was chosen to check strain limit. To apply Test No. B-2, two parameters, X and Y have to be obtained and they are defined as:

$$X = \frac{\left(P_L + \frac{P_b}{K_t}\right)_{max}}{S_{YL}} \quad (16)$$

$$Y = \frac{(Q_R)_{max}}{S_{YL}} \quad (17)$$

P_L is the local primary membrane stress, Q_R is the secondary stress range, S_{YL} is the minimum material yield strength at the minimum wall average temperature during the transient cycles, the subscript *max* means maximum value during the transient cycles to be evaluated, P_b and K_t are as defined before. After X , Y values are obtained, Z value can be evaluated according to the plots in Figure HBB-T-1332-2 in [2]. After getting Z value, the total strain during component service life ϵ_{total} can be obtained from HBB-T_1800 plots of [2] for a stress level of $1.25 * Z * S_{YL}$ and operation temperature T .

2.5.5 Creep-fatigue damage calculation procedure

Paragraph HBB-T-1432 of [2] gives different options for creep-fatigue damage calculation. The following creep-fatigue damage calculation procedure involves those options applied in this report, and should be regarded as paraphrase or excerpt of that paragraph.

2.5.5.1 Fatigue damage calculation

The fatigue damage can be calculated by following steps:

Step 1, check for ratcheting. To make fatigue damage calculation, it has to be ensured that no ratcheting occurs, and that can be avoided by checking the conditions:

$$Z < 1 \quad (18)$$

$$P_L + P_b + P_e + Q < \min(3S_m, 3\overline{S_m}) \quad (19)$$

Where Z is the parameter mentioned in Section 4.3, P_e is the pipe expansion stress, which is equal to 0 in our case, P_L , P_b and Q are as defined before, and

$$S_m = 0.5 * (S_{mL} + S_{mH}) \quad (20)$$

$$3\overline{S_m} = (1.5S_{mL} + S_{rH}) \quad (21)$$

Where S_{mL} and S_{mH} are S_m values at low temperature end and high temperature end for the Q calculation, respectively, S_{rH} is the relaxation strength at the hot extreme temperature of the stress cycle.

Step 2, strain range $\Delta\epsilon_{mod}$ calculations. The procedure is based on Equation (22),

$$\Delta\epsilon_{mod} = K_e K \Delta\epsilon_{max} \quad (22)$$

Where K is the stress concentration factor, equal to 1 in our case;

$\Delta\epsilon_{max} = \frac{2S_{alt}}{E}$, $2S_{alt}$ is the total stress intensity range from FEA results, E is the elastic modulus at maximum metal temperature during the cycle;

$K_e = 1$ if $K \Delta\epsilon_{max} \leq 3\overline{S_m}/E$;

$K_e = \frac{K \Delta\epsilon_{max} E}{3\overline{S_m}}$ if $K \Delta\epsilon_{max} > 3\overline{S_m}/E$.

Step 3, determine multiaxial plasticity and Poisson ratio adjustment factor K_v . K_v is defined in Equation (23),

$$K_v = 1.0 + f(K'_v - 1.0) \quad (23)$$

Where f is determined by Figure HBB-T-1432-2 of [2], according to larger values of triaxiality factors at two extremes of the stress cycle; K'_v is plastic Poisson ratio adjustment factor determined by Figure HBB-T-1432-3 of [2], based on the value of $\frac{K_e K \Delta\epsilon_{max} E}{3\overline{S_m}}$.

Step 4, Calculate the creep strain increment, $\Delta\epsilon_c$, which is equal to the creep strain at maximum metal temperature with stress intensity equal to $1.25*Z*S_{yL}$ for 700 hours, the steady state hold time of the operating transient.

Step 5, calculate total strain range ϵ_t according to Equation (24) below,

$$\epsilon_t = K_v \Delta\epsilon_{mod} + K \Delta\epsilon_c \quad (24)$$

Step 6, calculate allowable number of cycles N_d based on ϵ_t and temperature. Fatigue damage is the ratio of expected number of cycles to N_d . The fatigue data are from Figures HBB-T-1420-1B for SS316 from [2] and allowable number of cycles N_d can be obtained from related hbbdata data package functions^[5].

2.5.5.2 Creep damage calculation

There are two different calculation scenarios: when $\epsilon_t > 3\overline{S_m}/E$, the stress relaxation at high temperature is disturbed by stress resetting at low temperature, creep damage is calculated for each cycle, then the damages were summed up; when $\epsilon_t < 3\overline{S_m}/E$, no stress resetting at low temperature, and the stress relaxation at high temperature is not disturbed, and the high temperature holding during each cycles can be regarded as one holding through the whole service life. The calculations under both scenarios are conducted as damage accumulation through stress relaxation at holding temperature from a stress level S_{st} related to ϵ_t , and S_{st} values are given by Figure HBB-T-1800-B-11 hot tensile curves in [2], and stress levels S_i after relaxation for different time periods t_i are also provided there too in [2].

To calculate damage accumulation during stress relaxation, under the first scenario, within each stress cycle, we divided the high temperature holding time into N segments, and the i^{th} segment length is Δt_i , the starting stress level is S_i , the temperature is T_i , the creep damage during this segment is calculated through Equation (25) below and the starting stress level for the $i+1^{th}$ segment is obtained from HBB-T-1800-B-11^[2] at starting stress level S_i after relaxation time Δt_i . The creep damages from all time segments are summed up and that value multiplies the expected number of cycles will give the total creep damage during service life.

$$\Delta D_{c,i} = \frac{\Delta t_i}{T_{Si/0.9,Ti}} \quad (25)$$

Where $\Delta D_{c,i}$ is creep damage increment during i^{th} time segment, $T_{Si/0.9, Ti}$ is the rupture life at $S_i/0.9$ and T_i , rupture life values can be obtained from Figure HBB-I-14.6B of [2]. For the calculation of starting stress level for each time segment, two checks are needed:

1. During the first 20 hours for each stress cycle, whenever the starting stress level is lower than the stress intensity due to pressure stress, it is modified to that value and continue the stress relaxation process.
4. For any of the time segment, whenever the starting stress level is less than S_{LB} , it is changed to that value and the stress relaxation process continues. S_{LB} is defined as

$$S_{LB} = 1.25ZS_{yL} \quad (26)$$

Under second scenario, the whole service life is regarded as one stress cycle and only the second check is needed.

2.6 Assumptions

This analysis is based on the following assumptions:

- The additive manufactured SS316L, the material for the test article, is assumed to have the same properties as those for SS316H in [2].
- Materials are isotropic and homogeneous; material properties remain the same over time, i.e., effects of material degradation over time, are neglected.
- Residual stresses are not included and unnecessary for the elastic and simplified inelastic analysis according to [2].
- As the design is still in its conceptual stage and there is no detailed thermo-hydraulic analysis results ready for the test article operation. Very low temperature ramping rates (20 hours ramping time) were assumed in the flow transients and therefore no thermal stress peaks during the temperature ramping up or ramping down and the thermal stress variation can be calculated based on the stress analysis results at steady state. The assumed flow transient is shown in Figure 4 below. For 300,000 hours service life with the same flow transient, there will be 430 cycles.

Figure 4. Assumed flow transient applied in the analysis.

2.7 Analysis Inputs

2.7.1 Geometry and Mesh

The ANSYS model with heat-pipe microreactor geometry and boundary conditions was transmitted from LANL to ANL in December, 2019. When the model was prepared for FEA at ANL, the geometry was cut into half due to symmetry, so the new model represents 1/12 of the whole heat-pipe microreactor, instead of 1/6, as the original model did. The modified geometry is shown in Figure 2 of this report. The meshes for the core block, heat pipe and most of the heat exchanger were refined, and meshes for some parts of the heat exchanger with stress concentrations were refined through sub-modeling technique of ANSYS. Figure 5 displays meshes at different locations of the heat-pipe microreactor. There are total 0.5 million quadratic elements in the model.

Figure 5. Meshes in the heat-pipe microreactor FEA model. (a) core block, (b) heat exchanger, hot section, (c) heat exchanger, cold section. (d) heat pipes in the heat exchanger hot section, (e) heat pipes in the heat exchanger cold section.

The plan was to fabricate the test article that represents the core block and heat exchanger geometry of a heat-pipe microreactor by either 3D printing using Type 316L stainless steel or by the traditional route using wrought Type 316L stainless steel. It was assumed that the test article has similar properties as SS316H in the ASME code^[3]. The material properties applied in this report are provided by the hbbdata package, which is a material data package developed at Argonne in computer language Python^[5], to provide ASME BPVC Section III, Division 5 design data.

For design limit check using Elastic-Perfectly Plastic (EPP) method, the pseudo yield strength can be expressed as^[9]

$$\sigma_{y.pseud} = 0.87 S_0(720 \text{ }^{\circ}\text{C}) \quad (27)$$

Which is equal to 21.2 MPa; for service limit check, the pseudo yield strength can be expressed as

$$\sigma_{y.pseud} = 0.87 S_{mt}(T, t) \quad (28)$$

Where T is the local temperature, t is the expected service life. In FEA, the pseudo yield strength values used for service limit check are based on 300,000 hours life in the table below.

Table 1. Pseudo yield strength values for EPP analysis.

Temperature ($^{\circ}\text{C}$)	Pseudo yield strength (MPa)
500	92.0
550	75.3
600	44.2
650	26.8
700	15.6
750	7.8

2.7.2 Loads and Boundary Conditions

2.7.2.1 Mechanical loads and boundary conditions

Figure 6. Mechanical loads and boundary conditions. (a) constrains, (b) pressure load.

Figure 6 shows the mechanical loads and boundary conditions of the FEA model. The pressure load is 20 Barg, i.e., 200,000 Pa, on the heat pipe outside surface and heat exchanger inside surface. The pressures inside fuel rod holes and heat pips are negligible.

2.7.2.2 Thermal loads and boundary conditions

The thermal loads and boundary conditions are from [6] and the initial temperature of the model is 350°C . In the core block, as shown in Figure 7 the fuel rod inside walls are with heat flux

$22500\text{W}/\text{m}^2$ from the electrical heating, and the heat pipe inside walls are with temperature 700°C , all other surfaces are thermally insulated.

Figure 7. Thermal loads/boundary conditions in the core block.

In the heat exchanger part of the heat-pipe microreactor, thermal radiation occurs among the heat exchanger internal surfaces and heat pipe outside surfaces. Figure 8 shows the surface involved in radiative heat transfer. Research has been conducted for the convective heat transfer between the heat pipe outside surface and cooling medium and between the heat exchanger inside surface and the cooling media^[6]. Figure 9 shows the convective heat transfer parameters and involved surfaces.

Figure 8. Radiative heat transfer surfaces.

Figure 9. Convective heat transfer boundary conditions. (a) at heat pipe, (b) coordinate system used to evaluate h and T_0 , (c) at hot tube bank, (d) at cold tube bank.

As part of the efforts to reduce thermal stress, some of heat exchanger inside walls are thermally insulated and part of the heat exchanger outside wall is kept at 520 °C, those surfaces are shown in Figure 10(a) and (b) below, respectively.

Figure 10. (a) thermal insulation inside wall, (b) constant temperature at outside wall.

2.8 Calculation Results

The following presented calculation results, except those through EPP methods or for local check, are all based on elastic analysis which involves stress linearization and classification. As heat-pipe microreactor geometry is complicated, compared with conventional cylindrical container for pressure vessel stress analysis, it was divided into two parts for analysis, i.e., core block, heat pipes and heat exchanger, and for each part, stress linearization was conducted, and the maximum stress linear results for P_m , P_L , P_b and Q_R were chosen. Figure 11 shows stress linearization at different locations. The stress linearization procedures are explained in Section

2.5.1 of the report. For fatigue damage, total stress is needed and the locations were picked up based on total stress contour plot provided by ANSYS postprocessor.

Figure 11. Stress linearization locations at different parts of the heat-pipe microreactor. (a) core block, (b) heat pipe, (c) heat exchanger-hot, (d) heat exchanger-cold.

2.8.1 Design limit check

2.8.1.1 Elastic method

Based on assumed test article operation conditions, the design temperature and pressure are specified as 720 °C and 20 Barg. The stress linearization results at different locations in the heat-pipe

microreactor are shown in Table 2, it shows that Equation (1) and Equation (2) are satisfied at every locations and heat-pipe microreactor passes design limit check.

Table 2. Stress analysis results at different locations for design limit check.

Design condition: 720 °C/20Barg, unit: MPa

Location	P_m	S_0	$P_m < S_0$	P_L	P_b	$P_L + P_b$	$1.5S_0$	$P_L + P_b < 1.5S_0$
Core block	0.1	24.4	Yes	0.1	0	0.1	36.6	Yes
Heat pipe	15.3	24.4	Yes	15.3	0	15.3	36.6	Yes
HX, hot	7.2	24.4	Yes	9.6	0	9.6	36.6	Yes
HX, cold	7.2	24.4	Yes	9.8	0	9.8	36.6	Yes

2.8.1.2 Elastic-Perfectly Plastic (EPP) method

The Design Limit was also checked through EPP method, the detailed methodology was described in Section 2.5.2. Figure 12 displays the contour plot of the plastic strain, it shows most region of the heat-pipe microreactor does not yield, the solution is converged and the equivalent plastic strain is small ($<0.02\%$), heat-pipe microreactor passes design limit check through EPP method.

Figure 12. Contour plot of equivalent plastic strain for design limit check with EPP method.

2.8.2 Service limit check

The assumed flow transient for analysis was demonstrated in Figure 4 and it was assigned as Level A service load. The local temperatures were used for both the elastic and EPP design analysis.

2.8.2.1 Elastic method

Table 3 shows the results of service limit check at different locations of the test article using elastic method. Results demonstrate that the allowable service life is greater than 300,000 hours

and the usage factors, for primary membrane stress and primary membrane plus primary bending stresses, are less than 1 with large margins.

Table 3. Stress analysis results at different locations for service limit check.

Location	P_m (MPa)	P_t (MPa)	P_b (MPa)	Service temperature (°C)	Allowable service life (hours)	Usage factor (Eq.(3)/Eq.(5)) at 300,000 hours
Core block	0.1	0.1	0	713	>300,000	0.01/0.01
Heat pipe	15.3	15.3	0	700	>300,000	0.48/0.48
HX hot	7.2	9.6	0	695	>300,000	0.05/0.09
HX cold	7.2	9.8	0	525	>300,000	1.6E-5/2.1E-5

2.8.2.2 EPP method

Figure 13 is the equivalent plastic strain contour plot when the pseudo yield strength is based on material data with 300,000 hour service life. It shows the limit load solution is converged, most of the regions never yield and the walls between heating element holes yield through thickness, that shows the signs of possible local damages there, part of the heat pipe internal walls, stress concentration sites, i.e., some corners, at heat exchanger also yield. The equivalent plastic strain magnitudes at all locations are small ($<0.1\%$).

Figure 13. Equivalent plastic strain contour plot for service limit check.

2.8.3 Local check

Local check calculation procedure was described in Section 2.5.3. Figure 14 is the contour plot of damage parameter D at time $3.6E+7$ seconds, or 10,000 hours. On the plot, wherever $D \geq 1$, failure occurs there locally. The webs between heating element holes give D values of 0.95, which indicates those locations almost reach the end of their lives. Similar warnings were given in section 8.2.2 for the service limit check with EPP method but with 300,000 hours expected service life. Most regions of the core block and heat pipes shows D values of around 0.5, i.e., half of their lives, and heat exchanger shows less damage with D values at most regions are less than 0.3, the means 100,000 hour service consumes 30% of the total possible service life.

Figure 14. Contour plot of damage parameter D at 100,000 hours service life for local check.

2.8.4 Strain and deformation limit check

Strain and deformation limit check criteria was described in Section 4.3 and the calculation results are shown in Table 4. The data in the table show that the 300,000 hour service life strain limit ($<1\%$) are satisfied at core block, heat pipe and heat exchanger cold section, but heat exchanger hot section only meets 1% strain limit with 200,000 hour service life.

Table 4. Strain and deformation limit check.

Location	P_L (MPa)	P_b (MPa)	Q_{it} (MPa)	S_{yt} (MPa)	Holding temperature (°C)	K_t (MPa)	$X = (P_L + P_b / K_t) / S_{yt}$	$Y = Q_{it} / S_{yt}$	Z	Accu.Strain for 300K hours life
Core block	0.1	0	31.9	127	713	1.25	0.00079	0.25118	0.031	4.5E-5
Heat pipe	15.3	0	21.3	127	700	1.25	0.1205	0.1667	0.1415	8.15E-3
HX-hot	9.6	0	92.4	127	695	1.25	0.0756	0.7276	0.1665	1.37E-2/9.3E-3 (200K hours)
HX-cold	9.8	0	185.7	127	525	1.25	0.0772	1.4622	0.2599	3E-4

2.8.5 Creep-Fatigue damage check

For creep-fatigue damage check, four locations at core block, heat pipe and heat exchanger were chosen based on previous check results and total stress intensity contour plot. The locations are demonstrated in Figure 15 below.

Figure 15. Locations for creep-fatigue damage check.

The detailed calculation procedures were described in Section 2.5.5, the flow transient was demonstrated in Figure 4. Table 5 list the calculation results for fatigue damage calculations and Table 6 lists those for creep damage calculation.

Table 5. Fatigue damage calculation results.

Location	Total stress range (MPa)	Hot extreme temperature (°C)	Cold extreme temperature (°C)	E_H Elastic modulus at hot extreme temperature (MPa)	$\Delta\epsilon_{max}$	$\frac{3S_m}{E_H}$	K_f	$\frac{K_f K \Delta\epsilon_{max} E_H}{3S_m}$	K_{ϵ}	f	K_{σ}	$\Delta\epsilon_f$	Allowable number of cycles
Core block	31.9	700	350	1.40E5	2.28E-4	1.36E-3	1	1.67E-1	1	0.5	1	3.53E-5	283
Heat pipe	34.8	700	350	1.40E5	2.48E-4	1.36E-3	1	1.82E-1	1	0.26	1	2.18E-4	45
HX-hot section	691	695	350	1.41E5	4.91E-3	1.37E-3	3.59	1.29E1	1	1	1	2.85E-4	14
HX-cold section	1140	525	350	1.58E5	7.22E-3	1.87E-3	3.85	1.48E1	1	1	1	2.65E-4	12

Table 6. Creep damage calculation results.

Location	Ramping Time per cycle (hour)	Holding time per cycle (hour)	Holding Temperature (°C)	Load-control stress intensity S_L (MPa)	S_{LR} (MPa)	Num. of segments for stress relaxation calculation	Total creep damage
Core block	20	700	700	36.7	4.92	10000	0.0479
Heat pipe	20	700	700	65.1	22.5	10000	0.7379
HX-hot section	20	700	695	173.8	26.43	10000	35.798
HX-cold section	20	700	525	202.9	41.26	10000	9.680

For 300,000 hours service life with 430 cycles, based on allowable number of cycles shown in Table 5, the fatigue damage is greater than 1. If the creep damage and fatigue damage is computed for one cycle and Figure 4-2 creep-fatigue damage envelope is included, the allowable number of cycles is 3 cycles for the heat-pipe microreactor. Data in Table 7 supports the above conclusion.

Table 7. Creep-Fatigue damage calculation summary.

Location	Creep usage (1 cycle)	Fatigue usage (1 cycle)	Allowed creep usage	Allowed fatigue usage
Core block	1.12E-4	3.53E-3	1.12E-2 (100 cycles)	3.53E-1 (100cycles)
Heat pipe	1.72E-3	2.22E-2	6.88E-2 (40cycles)	8.88E-1 (40 cycles)
HX hot	8.35E-2	7.14E-2	2.51E-1 (3 cycles)	2.14E-1 (3 cycles)
HX cold	2.26E-2	8.33E-2	1.58E-1 (7 cycles)	5.83E-1 (7 cycles)

2.9 Discussion

The presented calculation is based on the procedures provided in ASME BPVC Section III, Division 5^[2] which include design limit check, service limit check, strain and deformation limit check and creep-fatigue damage check.

The design limit checks include primary load (pressure load) only and its goal is to guarantee safe operation under normal conditions, i.e., no structural collapse or pipe burst when loads are applied, and the current heat-pipe microreactor design passes the checks. Two methods for de-

sign limit check, i.e., elastic method and EPP method proposed in a Code Case draft^[3], were applied and they provided the same results. Stress linearization and categorization are unnecessary in EPP method, which is one of its advantage over elastic method.

The service limit check involves primary load only, the same as the design limit check does and it provides extra design margins for different service load levels. The results demonstrate that the current heat-pipe microreactor design will survive 300,000 hours service.

EPP method results for both the design limit check and service limit check show yield occurs at stress concentration sites in heat exchanger. Based on the definition of pseudo yield strengths ($0.87S_o$ for design limit check and $0.87S_{mt}$ for service limit check), those locations are prone to be damaged for a 300,000 hour service life and may raise issues on local fatigue damage. The yield at the walls between heating elements, shown by the service limit check through the EPP method, need redesign, as yield occurs cross wall, i.e., early creep damage most likely occurs in those regions across the walls.

The aforementioned EPP method provides satisfactory life assessment results for creep-ductile materials; for creep-brittle materials, in reality, the stresses are more difficult to redistribute during creep and the damages are more localized than creep-ductile materials, so, local-check is necessary. With material specific constitutive equations for creep (see Subchapter 5.2 for details), FEA results for local check show that for 100,000 hour service life the walls between heating elements show the maximum damage of 0.95 (≥ 1 , material is damaged), and the heat pipe show damage of about 0.5 and all other regions of the heat-pipe microreactor show damage less than 0.5. The reason for local check providing an estimated service life of 100,000 hours is that the model included creep damage attributed to thermal stress relaxation which was not included in either design limit check or service limit check which only involve pressure load (primary load). Stress probing at the wall between heating element holes shows stress relaxation and the thermal stress seems to exist through the whole service life, and it won't completely disappear through relaxation (Figure 16).

Figure 16. Total stress relaxation and comparison with pressure stress.

Plots in Figure 16 demonstrate part of the thermal stress could last through the service life, that implied it might be partially regarded as primary stress and the damage caused by thermal stress

should be included for service limit check. In order to get an upper bound of service life, thermal stresses were completely regarded as primary load, and the service limit check results are then shown in Table 8 below.

Table 8. Service limit check with thermal stress as primary load.

Location	P_m	P_L	P_b	Allowable service life (hours)	Service temperature ($^{\circ}\text{C}$)	Usage factor (membrane/membrane + bending)
Core block, fuel rod hole-fuel rod hole wall	31.9	31.9	0	5000	713	0.81/0.81
Heat pipe	14.5	18.8	0	23000	700	0.03/0.92 (at 23000 hours)
HX - hot	11.4	25.2	0	60000	695	0.03/0.81
HX - cold	17.9	65.8	0	>300,000	525	5E-5/0.01 (at 300,000 hours)

Compared with the data in Table 3, the results in Table 8 are more conservative, such as the walls between heating element holes, the usage factors are 0.81 at 5000 hour service life, while in Table 3, the usage factor there for 300,000 hour service life, the usage factor is only 0.01. Another comparison was made between the results from local check and those in Table 8, the local check involves stress relaxation through creep, the damage parameter D is 0.95 at the core block and about 0.5 at the heat pipes for 100,000 hour service life as shown in Figure 13, the results in Table 8 are still conservative, as the thermal stress there is completely regarded as primary stress and is not relaxed during creep.

For strain and deformation limit check, Test No B-2 in HBB-T-1332 of [2] was applied, this test is based on Sartory's work^[7], which includes peak thermal strain and continue to apply elastic core stress concepts of O'Donnell and Porowski^[8] but without restraints on geometry and load. The test requires $Z \leq 1$ and the design passes the check when the total creep ratcheting strain is less than 1% for the whole service life. For all the four locations checked, three of them are with creep ratcheting strain less than 1% for 300,000 hour service life, and one for 200,000 hour service life with a creep ratcheting strain of 0.935, that location is heat exchanger hot section. The elastic analysis results demonstrate there are high secondary stress (thermal stress) ranges Q_R at the heat exchanger (92.4 MPa at hot section and 185.7 MPa at cold section) in comparison with those at core block and heat pipes, because the temperature at heat exchanger hot section (695 $^{\circ}\text{C}$) is much higher than that at cold section (525 $^{\circ}\text{C}$), the creep ratcheting strain there is more than 1% (1.31%) for 300,000 hours service life.

The elastic core stress concept was also applied to creep-fatigue damage calculations. Creep strain increment $\Delta\epsilon_c$, which is the creep ratcheting strain during one stress cycle and a function of elastic core stress at holding temperature, was included in total strain ϵ_t calculation (see Section 5.4.1 for details) to estimate allowable number of fatigue cycles based on material property data. During creep damage calculation, the initial stress is based on ϵ_t , and during stress relaxation, the relaxed stress intensities are compared with S_{LB} , which is also a function of elastic core stress (see Section 5.4.1 for details).

2.10 Analysis summary

The test article design was checked through the ASME BPVC Section III, Division 5 code. The checks include design limit check, service limit check, strain and deformation limit check and

creep-fatigue damage checks. While applying elastic analysis methods, stress results at different locations from FEA model were linearized and compared with those required by the checks aforementioned; for strain limit check, there are different ways for strain and deformation limit check in [2] and for our cases, Test No B-2 was chosen, as there is no restrictions on load and geometry for its application and it provides the parameter Z values for creep-fatigue damage calculations. In addition to elastic methods applied for those checks and as part of the code development efforts^[3], elastic-perfectly plastic methods, recommended in a Code Case draft, were also applied to design limit check and service limit check, and local check with material-specific creep constitutive equations was also conducted and provided estimated service life of 100,000 hours, which is more conservative than 300,000 hour service life provided by design limit check and service limit check with elastic method. With the restraint of the total strain $< 1\%$, the strain and deformation limit check demonstrates a service life of 200,000 hours. Creep-fatigue damage check based on a 20 hour ramping time and 700 hours holding at high temperature provides a life of 3 cycles, i.e., after 3 stress cycles of to total 2100 hours service. This check indicates the test article would not be a suitable design for an actual reactor designed to see long-term service.

3 Conclusion

This report evaluates the LANL test article design against the Section III, Division 5 Class A component design criteria. The general conclusion is that the article would have a very high maximum design life when only considering primary loads (i.e. pressure) but has a much lower design life when considering secondary, thermal stresses and, in particular, the Code creep-fatigue design criteria.

Thermal stresses are the cause of the low creep-fatigue life. The pressures in the reactor are essentially negligible, but the thermal stresses are quite high, particular in two key locations:

- The wall in the core block separating the heating elements (which would be the wall separating fuel elements in a corresponding heat-pipe microreactor design).
- Stress concentrations in the corners of the heat exchanger.

Creep and cyclic plasticity will tend to reduce these initially high thermal stresses through stress relaxation. However, a steady-state stress relaxation analysis of the component (Figure 16) demonstrates that the thermal stresses are so high that they will remain significant over any reasonable component design life. Cyclic loading, which is expected for heat-pipe microreactor designs, will make this problem more severe by potentially resetting the initial high thermal stress during each load cycle. The steady relaxation analysis shown in Figure 16 is to some extent a best-case scenario.

Considering an actual heat-pipe microreactor based on this test article, the plant thermohydraulics could be optimized to reduce the thermal stresses imposed on the core block and heat exchanger. This is especially true for the transient loading conditions used here in the creep-fatigue evaluation. Structurally, reducing the fuel hole spacing (i.e., using a thinner wall) would reduce the secondary stress at this key location. The authors are, however, aware this would impact the reactor neutronics. The fillet size and geometric design of the heat exchanger could likewise be tuned to reduce the limiting values of thermal stress.

The ASME Section III, Division 5 rules are not optimized for the design of components with high secondary stresses relative to the primary stresses. The Code rules were developed for larger advanced reactor designs operating at higher pressures and predominately exercised on the Clinch River sodium-cooled fast reactor design. That said, the complete Class A design approach remains adequately conservative for a thermal-stress dominated reactor design – the creep-fatigue design criteria will capture the damaging effect of high, resetting thermal stresses. However, these rules are the most complicated in the Division 5 design process and typically designers consider these checks last. Moreover, iterating through different geometric designs requires redoing the transient stress analysis for the strain accumulation and creep-fatigue checks and so historically designers used the primary load allowable stress criteria to size components.

The draft EPP primary load design rules [9] evaluated here alleviate this practical design challenge by accounting for the thermal stresses earlier in the design process. Note, however, that

the draft rules have not been approved by ASME. A nuclear Code Case is currently being considered that would allow the use of the EPP primary load design method in place of the current allowable stress rules.

Further optimization of the design rules to better accommodate structures controlled by thermal stress is possible. For example, simplified, more accurate methods for creep-fatigue design based on a Simplified Model Test (SMT) approach are being developed at Argonne National Laboratory and Oak Ridge National Laboratory [10-11]. These rules simplify the process of creep-fatigue design, potentially allowing designers to more quickly iterate through geometric designs while considering creep-fatigue damage. Moreover, this new approach may be more accurate and less over conservative than the current Code rules, allowing for more efficient structural designs.

ACKNOWLEDGEMENTS

The research was sponsored by the U.S. Department of Energy, under Contract No. DEAC02-06CH11357 with Argonne National Laboratory, managed and operated by UChicago Argonne LLC. Programmatic direction was provided by the Office of Nuclear Reactor Deployment of the Office of Nuclear Energy. The authors gratefully acknowledge the support provided by Sue Lesica, Federal Manager, Advanced Materials, Advanced Reactor Technologies (ART) Program, Tom Sowinski, Federal Manager, Microreactor Program, and Jess C. Gehin of Idaho National Laboratory, National Technical Director, Microreactor Program.

Discussions with Holly Trellue, Robert Reid, Lindsey Gaspar and Katrina Sweetland of the Los Alamos National Laboratory are also gratefully acknowledged.

REFERENCES

- [1] *Special Purpose Nuclear Reactor (5MW) for Reliable Power at Remote Sites Assessment Report: Using Phenomena Identification and Ranking Tables (PIRTs)*, INL/EXT-16-40741, Idaho National Laboratory, April 2017.
- [2] *ASME Boiler and Pressure Vessel Code 2019, Section III, Division 5*, American Society of Mechanical Engineers, Little Falls, NJ, 2019.
- [3] Personal communication between G. Ye and M.C. Messner, December 2019
- [4] Bree, J., “Elastic-Plastic Behaviour of Thin Tubes Subjected to Internal Pressure and Intermittent High-Heat Fluxes with Application to Fast-Nuclear-Reactor Fuel Elements,” *Journal of Strain Analysis for Engineering Design*, Vol 2, Issue 3, 1967
- [5] *HBBDATA Documentation, Release 0.1*, Argonne National Laboratory, August 2019.
- [6] *Heat Exchanger Design and Analysis*, LA-UR-19-29276, Los Alamos National Laboratory, September 2019.
- [7] Satory, W.K., “Effect of peak thermal strain on simplified ratchetting analysis procedures,” *Joint ASME/JSME pressure vessel and piping conference*, Honolulu, HI, USA, 23 July 1989.
- [8] Porowski, J.S., O'Donnell, W.J., “Creep ratcheting bounds from extended elastic core concept,” *International conference on structural mechanics in reactor technology*, Berlin, F.R. Germany, 13 August 1979.
- [9] Messner, Mark C., and Sham, T.-L. *Initial development and verification of a primary load design method based on elastic-perfectly plastic analysis*. Technical report, Argonne National Laboratory, Lemont, IL, ANL-ART-201, 2020
- [10] Wang, Y., Tianlei, L., Sham, T.-L., and Jetter, R. I. “Evaluation of Creep-Fatigue Damage Based on Simplified Model Test.” In *Proceedings of the ASME 2013 Pressure Vessels & Piping Division Conference (PVP2013-97113)*, 2013.
- [11] Messner, M. C., Jetter, Robert I, Wang, Yanli, and Sham, T. -L. *Initial Development of an Improved Creep-Fatigue Design Method that Avoids the Separate Evaluation of Creep and Fatigue Damage and Eliminates the Requirement for Stress Classification*. Technical report, Argonne National Laboratory, Lemont, IL, ANL-ART-168, 2019.

Applied Materials Division

Argonne National Laboratory
9700 South Cass Avenue, Bldg. 212
Argonne, IL 60439

www.anl.gov

Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC