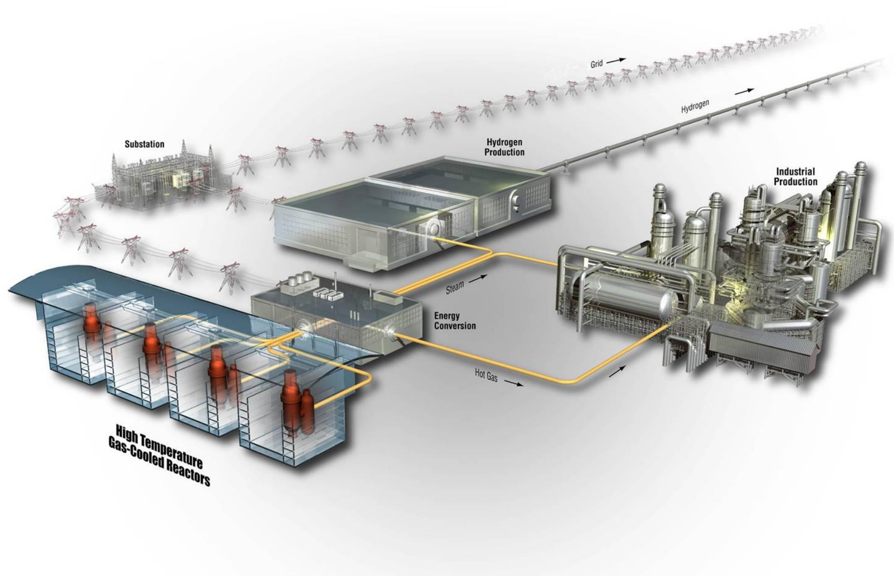
INL/EXT-12-26626

HTTF Core Stress Analysis

Brian D. Hawkes
Richard R. Schultz

July 2012



The INL is a
U.S. Department of Energy
National Laboratory
operated by
Battelle Energy Alliance

Idaho National Laboratory

INL/EXT-12-26626

HTTF Core Stress Analysis

Brian D. Hawkes
Richard R. Schultz

July 2012

Idaho National Laboratory
VHTR TDO Program
Idaho Falls, Idaho 83415

http://www.inl.gov

Prepared for the
U.S. Department of Energy
Office of Nuclear Energy
Under DOE Idaho Operations Office
Contract DE-AC07-05ID14517

VHTR TDO Program

HTTF Core Stress Analysis

INL/EXT-12-26626

July 2012

Approved by:

Brian Hawkes by RRS

Brian D. Hawkes
Author

7/26/2012
Date

Richard R. Schultz

Richard R. Schultz
PI

7/26/2012
Date

Diane V. C.

Diane Croson
VHTR TDO Deputy Director

7/26/12
Date

ABSTRACT

This study is based on the need to evaluate cracking of the ceramic core disks to be constructed and used in the High Temperature Test Facility (HTTF) at Oregon State University for heatup and cooldown experiments. A set of calculations was performed using Abaqus to investigate the thermal stresses levels and likelihood for cracking. The calculations showed that using the material properties provided for the GreenCast 94F ceramic, cracking is predicted to occur. However, this modeling does not predict the size or length of the actual cracks. It is quite likely that cracks will be narrow with rough walls which would impede the flow of coolant gases entering the cracks.

Based on data recorded at Oregon State University using GreenCast 94F samples that were heated and cooled at prescribed rates, it was concluded that the likelihood of cracks being detrimental to experimental objectives is low.

CONTENTS

FIGURES

TABLES

HTTF Core Stress Analysis

1. INTRODUCTION

A concern was raised about the structural viability, at operating temperature, of the ceramic material that will be used for the core in the High Temperature Test Facility (HTTF) at Oregon State University. The thermal stresses imposed on the ceramic will actually occur because of temperature gradients. Such gradients will occur between the high temperatures of the heat rods and the lower temperatures of the coolant channels, the center reflector, and the outer walls. While higher temperature gradients may be possible during the heat-up prior to a steady-state test, it was decided to first perform a stress analysis on the ceramic material at its projected steady-state operating temperature.

The design and layout of the coolant and bypass channels in the HTTF core have undergone a series of reconfigurations. Though some relatively minor modifications may still occur to the design of the core, the configuration on which the presently reported calculations are based should be close to the final design. It was determined to perform the stress calculations using a commercial finite-element stress analysis code. However, it was further determined that the temperature distribution would be computed using a conjugate flow and heat transfer analysis with a commercial computational fluid dynamics (CFD) code. Upon completing the CFD analysis, the calculated temperatures would then be transferred for use by the stress analysis code. The mesh used in the stress code, though different than that used in the CFD code, has about the same mesh density. The CFD code is used to map or interpolate the temperature distribution onto the stress mesh as described below.

2. CFD MODEL

The temperature distribution of the ceramic is needed to perform stress analyses on the thermally stressed ceramic in the HTTF core. The temperature of the HTTF core for steady-state conditions was therefore computed for the conjugate fluid flow and heat transfer problem using a fine-grid computational fluid dynamics (CFD) model. The calculations were made using the commercial CFD code STAR-CCM$^+$ [1]. Temperature dependent properties were used for the helium coolant, and the material properties of ceramic GreenCast 94F were used for the core. These properties were supplied by the manufacturer. While the CFD model does not include the permanent side reflector, information from a RELAP5 [2] calculation, which did include the side reflector, was used for the outer temperature boundary condition. For the RELAP5 model, properties used for the permanent side reflector are those for ceramic Thor 80. The configuration for the HTTF core was Revision E, with an overall hexagonal shape for the core (excluding the permanent side reflector).

The CFD model was run iteratively, adjusting the pressure drop across the core until the overall mass-averaged outlet temperature was approximately 960 K, based on a 528 K inlet temperature. Properties for GreenCast 94F and the heater rod material used in the CFD model were as follows:

- Properties for GreenCast 94F:

  density = 2930 kg/m^3
  heat capacity = 1200 J/kg K
  thermal conductivity = function of T (W/m K) as shown in Figure 1.

- Material properties used for the heater material:

  density = 1800 kg/m^3
  heat capacity = $4.041 \times 10^{-14}\,T^5 - 4.464 \times 10^{-10}\,T^4 + 1.964 \times 10^{-6}\,T^3 - 0.004353\,T^2 + 5.062\,T + 471$ J/kg K
  thermal conductivity = $0.0166\,T + 14.204$ W/m K.

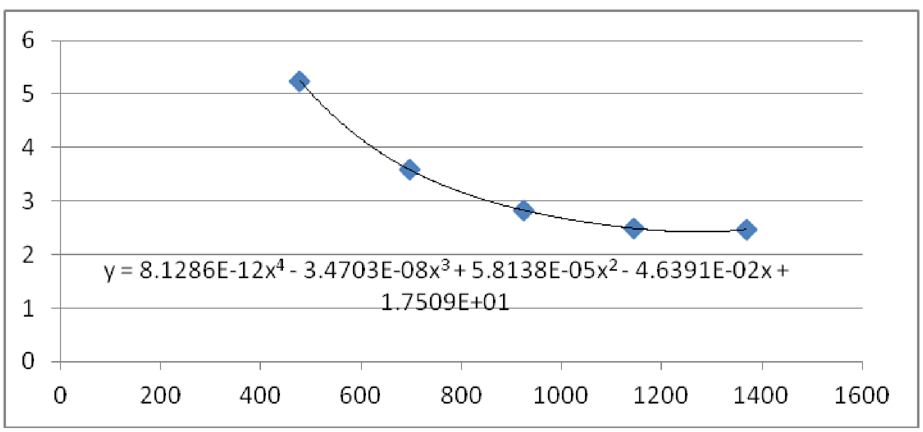


Figure 1. Thermal conductivity (W/m K) of GreenCast 94F as a function of temperature (K).

Details for the CFD model were as follows:

- All heater rods have the same power (1.8443×10^7 W/m^3, 2.188 MW total, 210 heater rods).
- The axial power profile is flat.
- The coolant used in the model is helium, with properties (obtained from NIST [3]) at 700 kPa. The properties for density, thermal conductivity, and dynamic viscosity are fitted to curves that are used in the CFD code. The properties are valid from 300 to 2200 K.

- The helium inlet temperature is set to 528 K.
- The inlet boundary condition is set to stagnation pressure at 1370 Pa; the outlet pressure is set to 0 Pa. To obtain the absolute pressure, a reference pressure of 698.552 kPa must be added to the above pressures. The pressure losses at coolant channel inlets and outlets are part of the solution.
- A one-twelfth sector of the HTTF core is modeled with symmetry boundary conditions on the sides of the model. Figure 2 shows the layout of the CFD model with the upper reflector toward the upper left, the heated core in the middle, and the lower reflector toward the lower right. Figure 3 provides a cross sectional view of the CFD model indicating the locations of the inner and outer bypass channels (which function as coolant channels), the normal coolant channels and the heater rods, each surrounded by a narrow annulus filled with stagnant helium.
- The outer boundary is based on temperatures computed from a REPLAP5 model, which includes the permanent side reflector. Hence, the inner surface of the permanent side reflector represents the outer surface of the CFD model. The axial (vertical) temperature distribution computed by RELAP5 and used for the outer boundary condition for the CFD model is shown in Figure 4.
- The inlet and outlet faces of the core are adiabatic.
- There are continuous flow paths through the HTTF, with no junctions between disks.
- The initial 0.2974 m of core is unheated (upper reflector).
- The next 1.9825 m of core is heated.
- The last 0.4 m of core is unheated (lower reflector).
- The realizable two-layer k~ε turbulence model with the two-layer all y^+ wall treatment are used for the coolant flow in the model. This turbulence model has produced excellent results for wall shear stress and heat transfer coefficient for turbulent flow in smooth tubes [4].
- The annuli between the 0.75 in. dia. heater rods and the 1.0 in dia. holes in the core that contain the heater rods are assumed to be filled with stagnant helium.
- The inner bypass channel is 0.75 in. in diameter.
- The three outer bypass channels are 0.625 in. in diameter.
- The coolant channels surrounding the heater rods vary from 0.375 in. to 0.5 in. to 0.625 in.
- The CFD mesh has about 18.1 million cells.

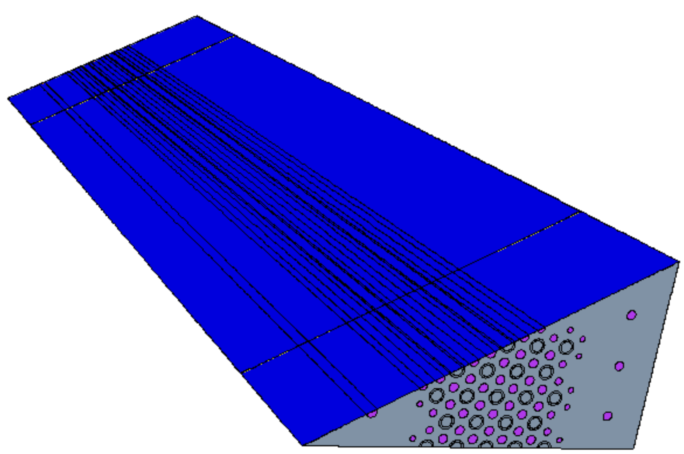


Figure 2. Illustration of the CFD model for the 1/12 sector.

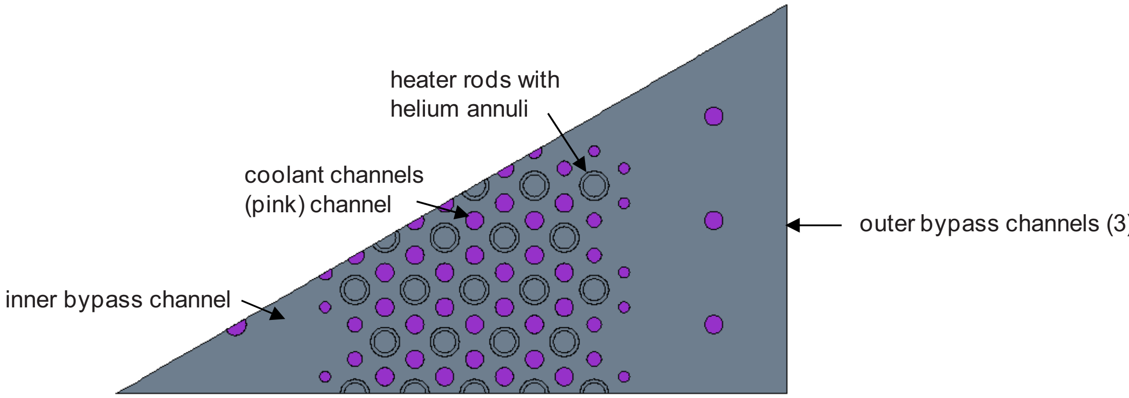


Figure 3. Cross sectional view of the CFD model.

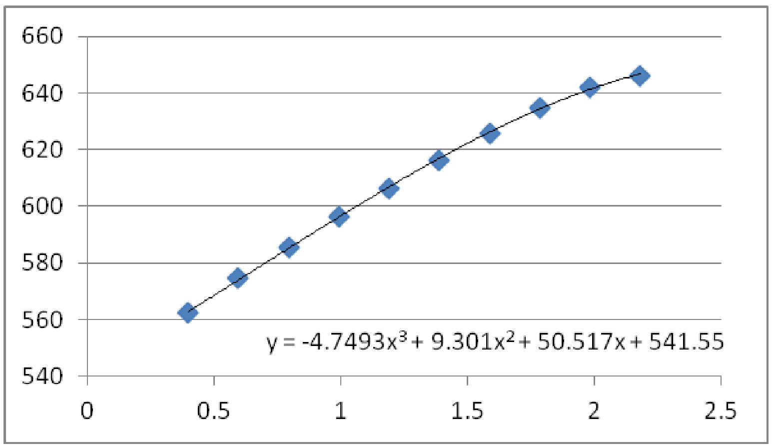


Figure 4. Temperature (K) distribution along the vertical edge of the permanent side reflector (m) (temperature increasing with core depth as measured from the top).

3. RESULTS FOR TEMPERATURE DISTRIBUTION IN HTTF

The mass flow and temperatures of the simulation are given in Tables 1 and 2 are compared to those of the RELAP5 calculation. Figure 5 illustrates the temperature contours in a plane in the ninth disk, which is the hottest disk in the core. Figure 6 shows a close-up view of the temperature contours in the ceramic only and the mesh. The larger circles are the locations of the heaters; the smaller circles are the locations of the coolant channels. The close-up is taken from the approximate center of the CFD model. Note that it is the temperature *gradient* that creates the thermal stresses.

The temperature field computed by STAR-CCM+ is then transferred to stress analysis code Abaqus/Standard [5]. This is accomplished by reading an input file from Abaqus/Standard that contains mesh information for the Abaqus/Standard mesh. The temperatures are then mapped onto the Abaqus/Standard grid from the CFD grid and output to a file that can be read by Abaqus/Standard. Figure 7 plots the temperature computed by the CFD code compared to that which has been remapped (interpolated) onto the Abaqus/Standard mesh, giving an indication of the accuracy of the remapping procedure.

Table 1. Mass flows and exit temperature for the HTTF coolant channels for CFD and RELAP5 results.

Parameter	Inner Bypass		Outer Bypass		Main Coolant Channels	
	RELAP5	CFD (x12)	REALP5	CFD (x12)	RELAP5	CFD (x12)
Mass flow (kg/sec)	0.01712	0.02701	0.09868	0.10048	0.84983	0.838305
Exit Temp (K)	728.4	665.9	630.4	640.6	1003.0-1014.5	1007.5

Table 2. Overall mass flows and temperatures for the CFD results compared to RELAP5 results.

Parameter	Total mass flow (kg/sec)	Inlet temperature (K)	Mass average outflow temp (K)	Total energy input (MW)
RELAP5	0.96545	528.8	960.15	2.2
CFD	0.9658	528	959.79	2.188

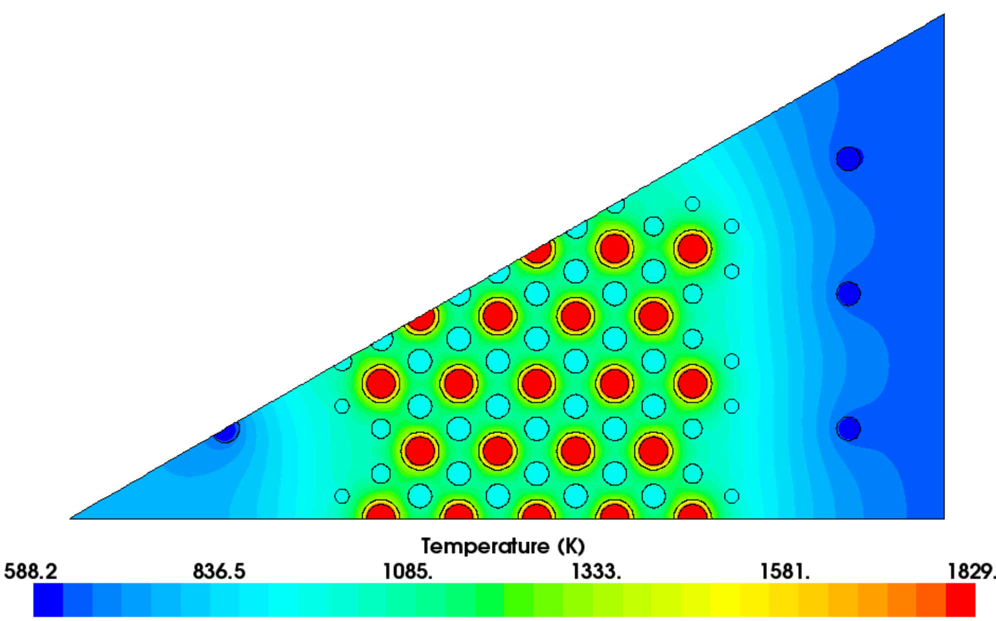


Figure 5. Temperature contours in the plane of the ninth lowest disk.

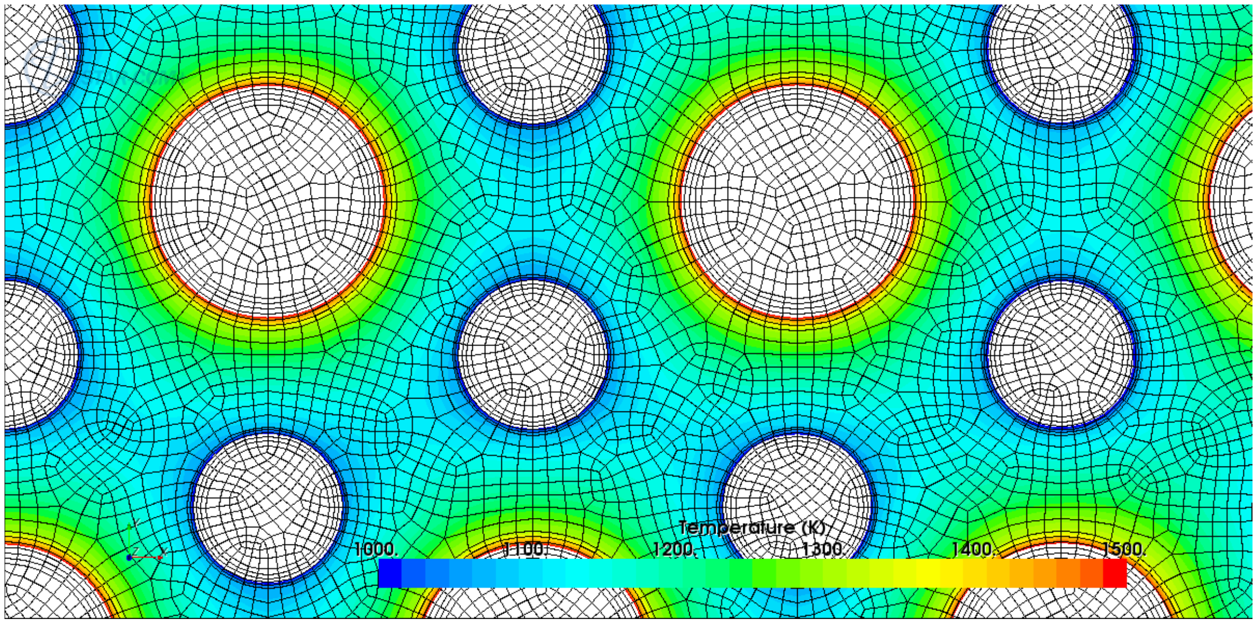


Figure 6. Close-up view of temperature contours in the ceramic core and the mesh for the CFD result.

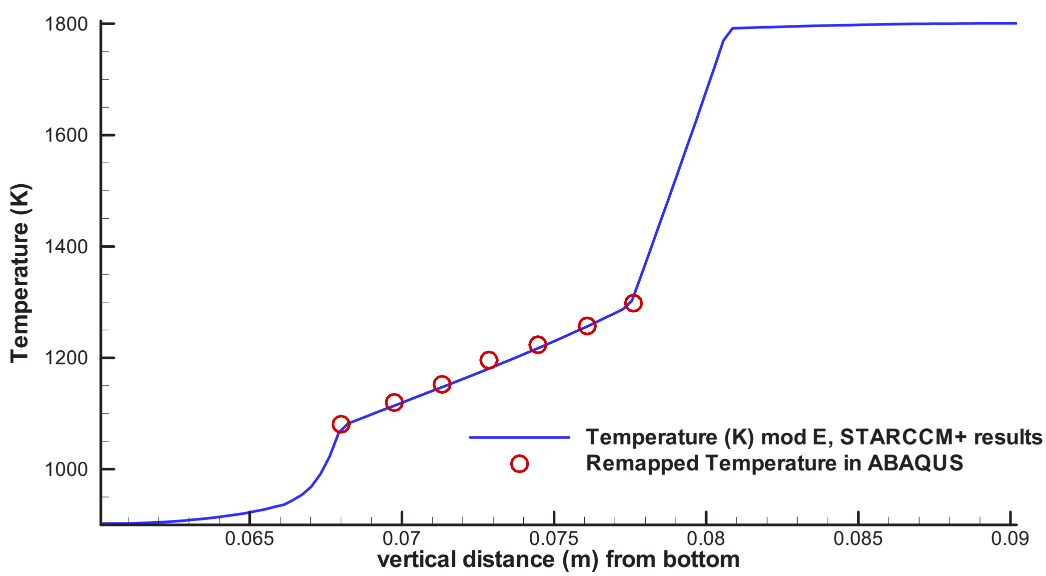


Figure 7. Comparison of temperature computed using CFD code to that remapped onto the stress analysis mesh.

4. THERMAL STRESS ANALYSIS

A finite element model of a section of the ninth disk was created using I-DEAS NX6 [6]. This finite element model (Figure 8) uses standard brick or hexahedral elements. The model was meshed with a density that approximated the STAR-CCM+ mesh that was used to obtain the temperature distribution for the ceramic core. The elements have an aspect ratio of about 8:1 in the Z or axial direction, since the temperature gradient is very gradual in this direction. This finite element model was converted to an Abaqus/Standard input file and solved for thermal stresses resulting from thermal gradients.

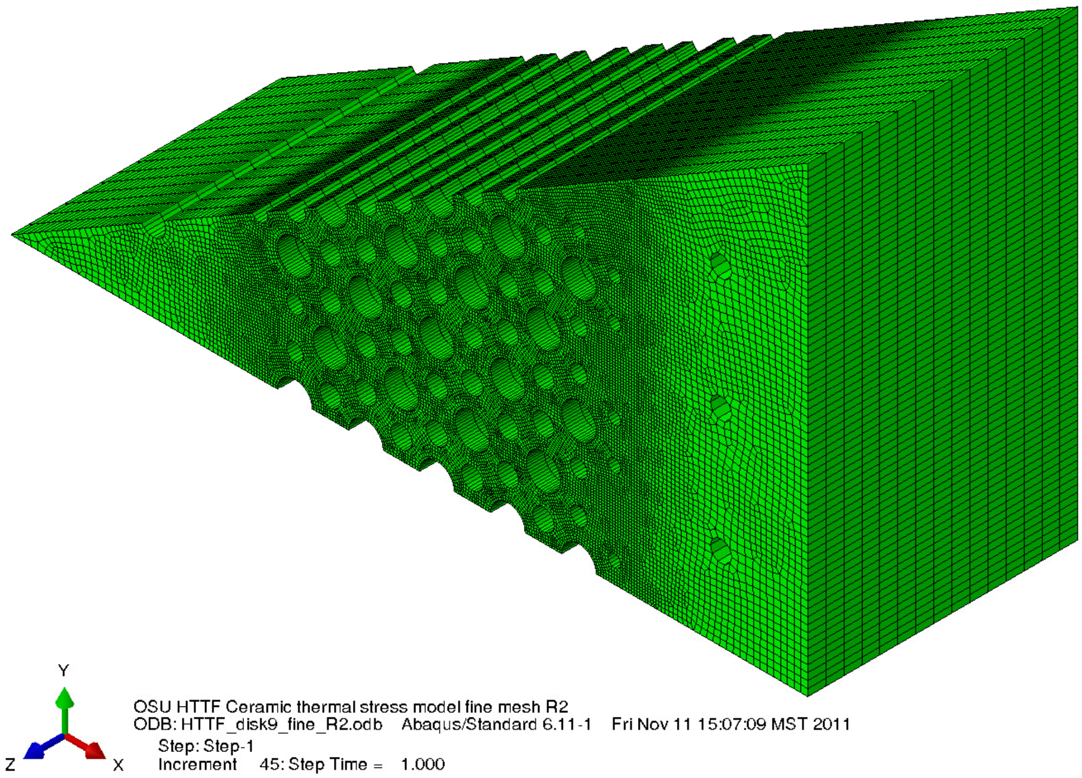


Figure 8. Abaqus/Standard finite element mesh of a section of ninth disk.

This model of one-twelfth of the disk uses symmetry boundary conditions to represent the entire disk. The bottom face was restrained from movement in the Y-direction. (See Fig. 8 for the coordinate system orientation). The whole part was restrained from movement in the Z-direction by weak springs in each of the three corners. All of the nodes on the slanted upper face were placed in a transformed coordinate system where the transformed X-axis runs along the corner of the upper face and the front face of the model and the transformed Y-axis is perpendicular to the upper face. The Z-axis remained unchanged. Using this transformed coordinate system, the nodes on the upper face were restrained from movement perpendicular to the face. This restrained the model from movement in the X-direction while allowing radial thermal expansion to occur.

The input file for this finite element model was used to map the temperatures from STAR-CCM+, as described above. STAR-CCM+ returned a file that mapped the calculated temperatures to the nodes in this model as shown in Figure 9. Figure 10 shows a close-up view of the temperatures in the center part of the model.

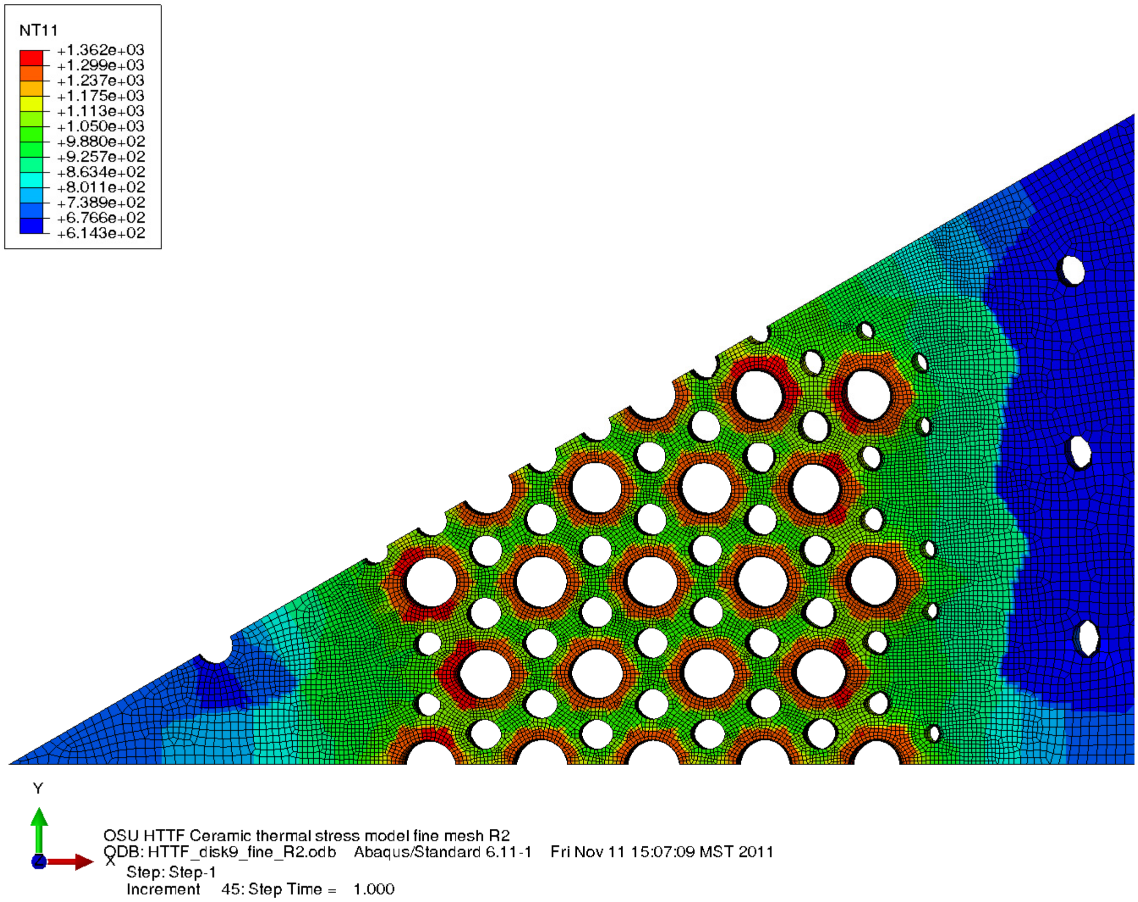


Figure 9. STAR-CCM+ model temperatures (K) mapped onto the Abaqus/Standard model.

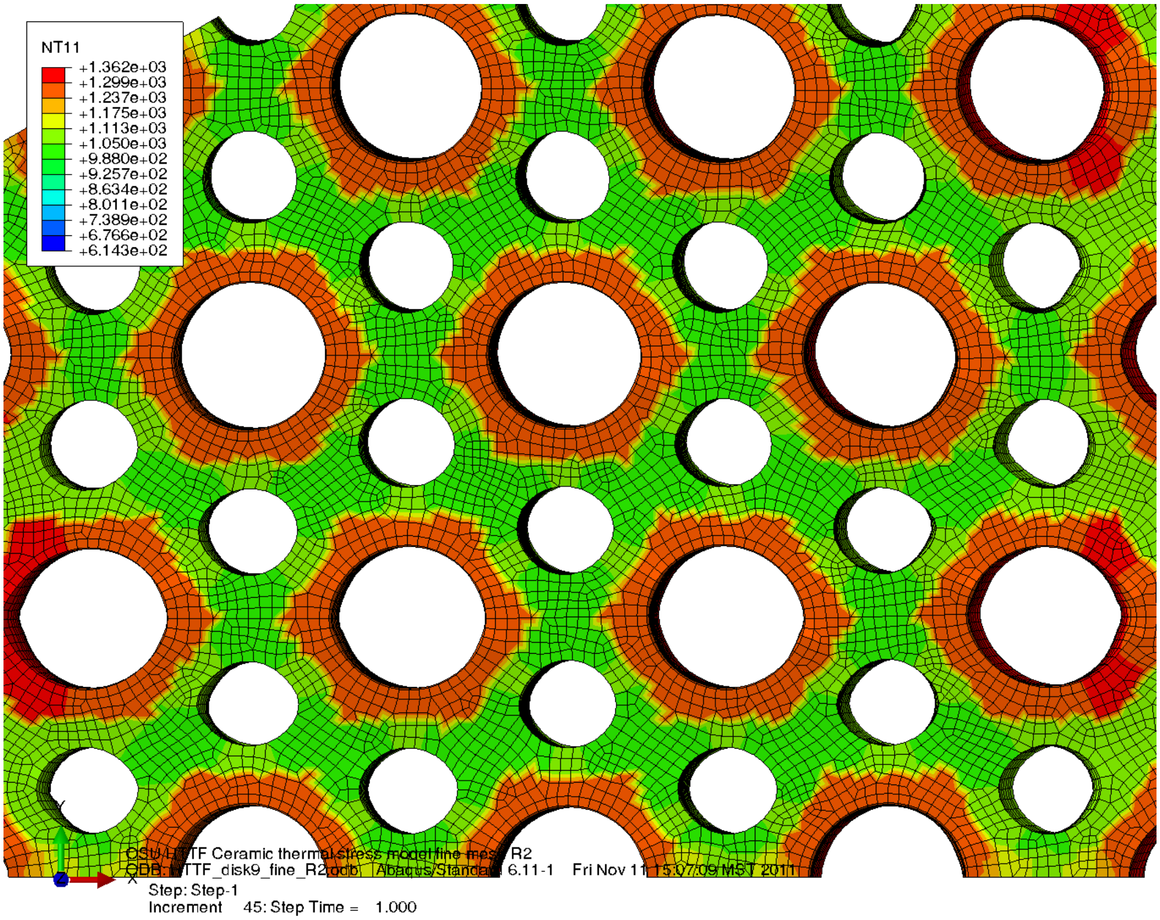


Figure 10. Close-up view of the mapped temperatures (K) in the Abaqus/Standard model.

4.1 Material Properties

The GreenCast 94F cast ceramic material was modeled as an elastic-perfectly plastic material, meaning that the material was treated as a linear elastic material until the yield strength was reached and then it was perfectly plastic. This permitted the elements that exceeded the yield strength to be displayed with plastic strain (PEEQ) as shown in Figure 13. The material properties given in Table 3 and Figure 11 were furnished by B. Jackson (OSU) for use in this analysis:

Table 3. GreenCast 94F material properties

Material Property	Value		Units
Modulus of elasticity	4.2265E+10		Pa
Poission's ratio	0.090		Unitless
Density	2912.6		Kg/m^3
Thermal expansion	See Figure 11		1/K
Yield strength	17.2	1089 K	MPa
	19.3	1644 K	<Pa
	23.4	1755 K	MPa

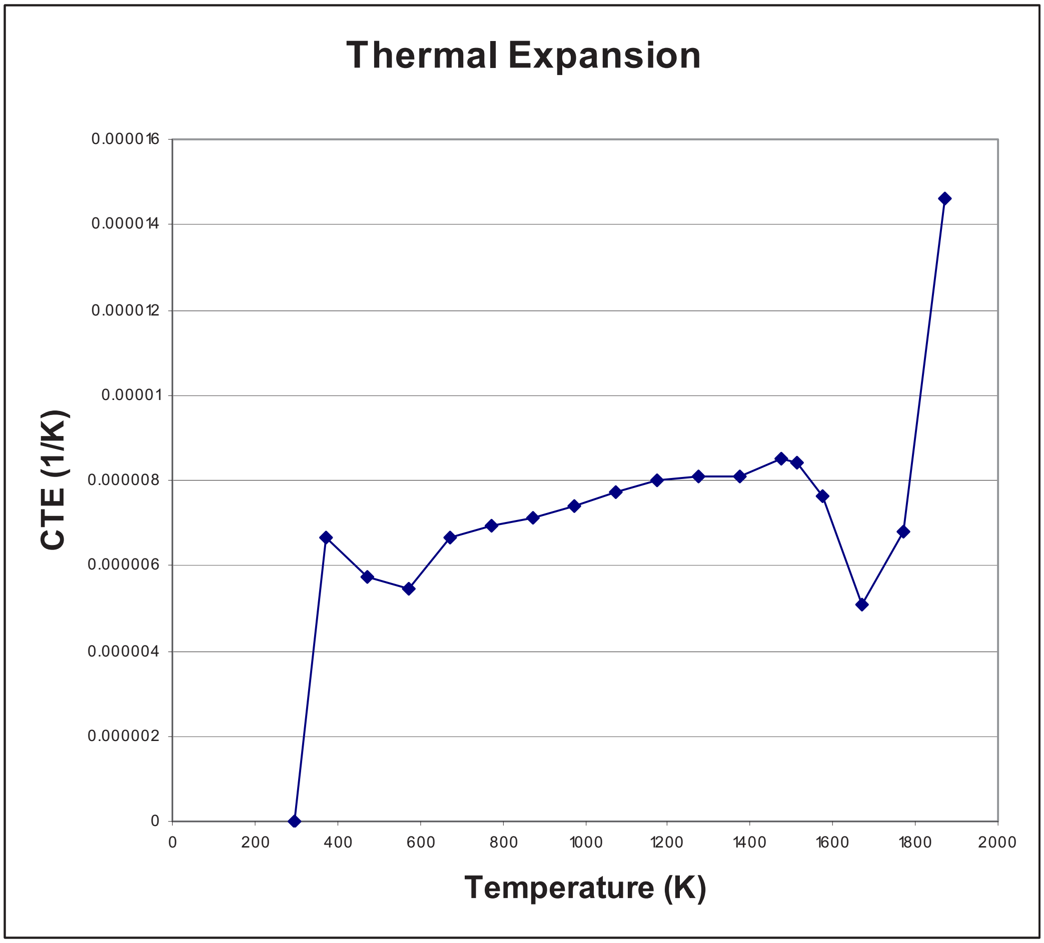


Figure 11. Measured coefficient of thermal expansion for GreenCast 94°F.

5. RESULTS

Results of the thermal stress analysis using this model are shown in Figures 12 and 13. The calculated von Mises stress is shown in Figure 12. The stresses shown in Fig. 12 can be compared to the yield stresses in Table 3. The maximum stresses exceed the yield strength, which shows that plastic strain occurs in the model. Figure 13 illustrates the plastic strain. Wherever the plastic strain is above zero, the material has exceeded the yield strength calculated for that temperature and cracking is predicted to occur. Generally, these predicted areas of cracking occur between the heater rod holes and the coolant channels. Figure 14 shows all elements where any plastic strain occurs. Exceeding the yield strength is an indication of cracking for brittle materials. However, it is expected that a small amount of plastic strain would occur in the ceramic before the onset of cracking. This would reduce the amount of cracking in the model. Figure 15 shows the calculated deformation in the model. The maximum movement (radially) is just over 3 mm.

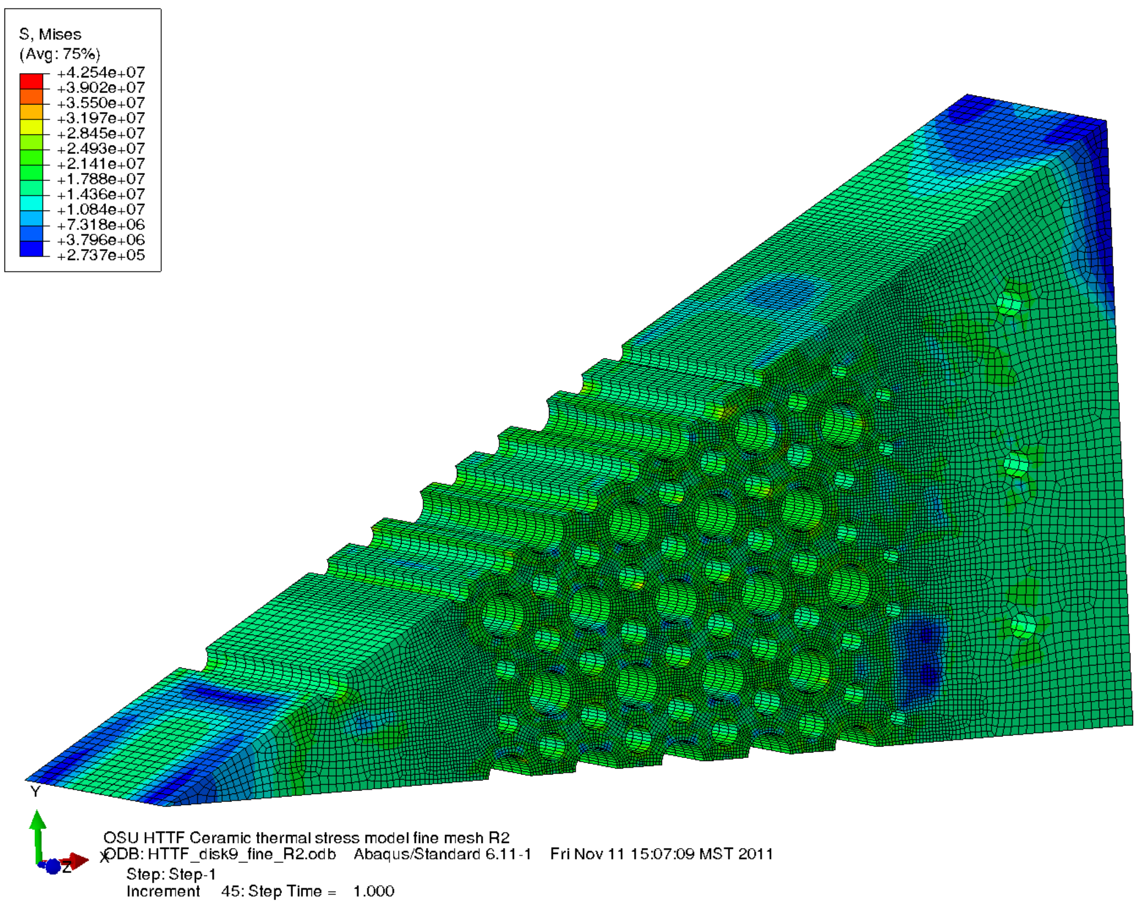


Figure 12. Calculated von Mises stress (Pa) in the GreenCast 94F ceramic.

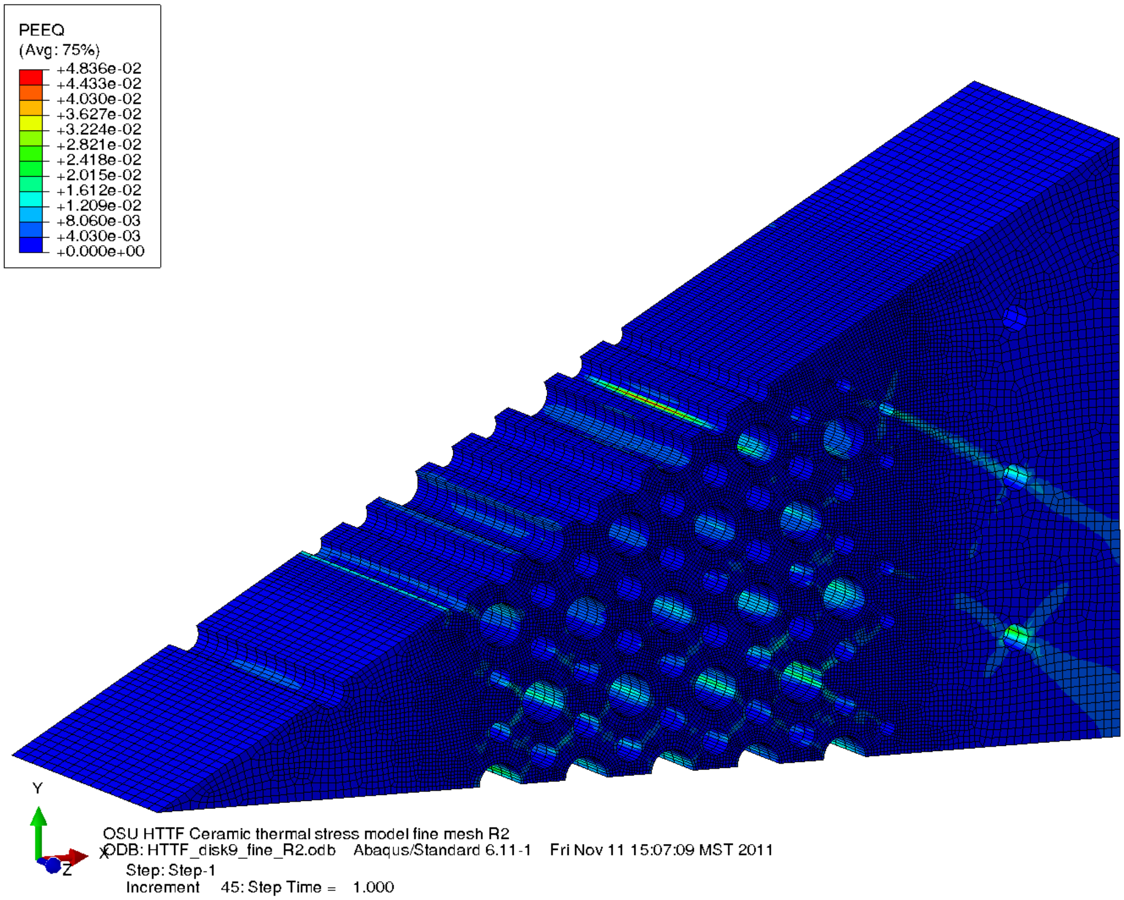


Figure 13. Calculated plastic strain from thermal expansion.

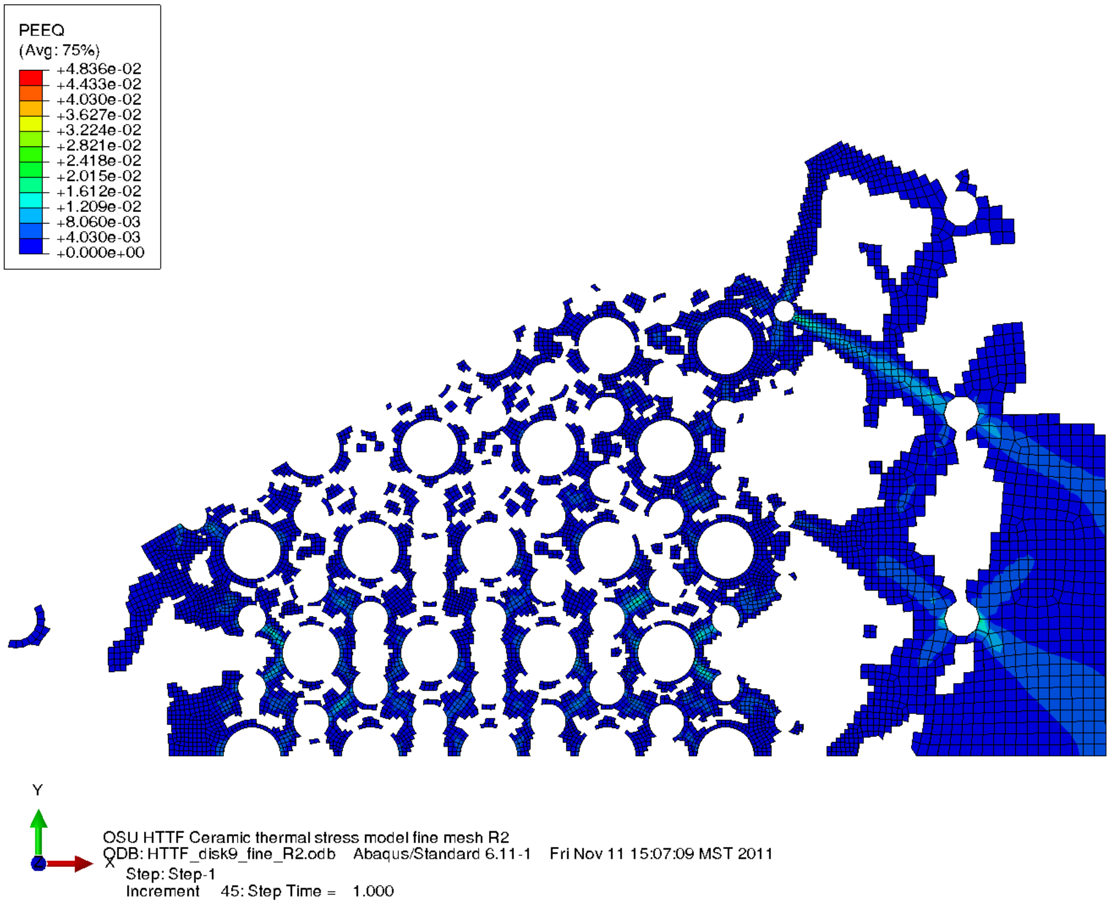


Figure 14. Elements exhibiting some degree of plastic strain.

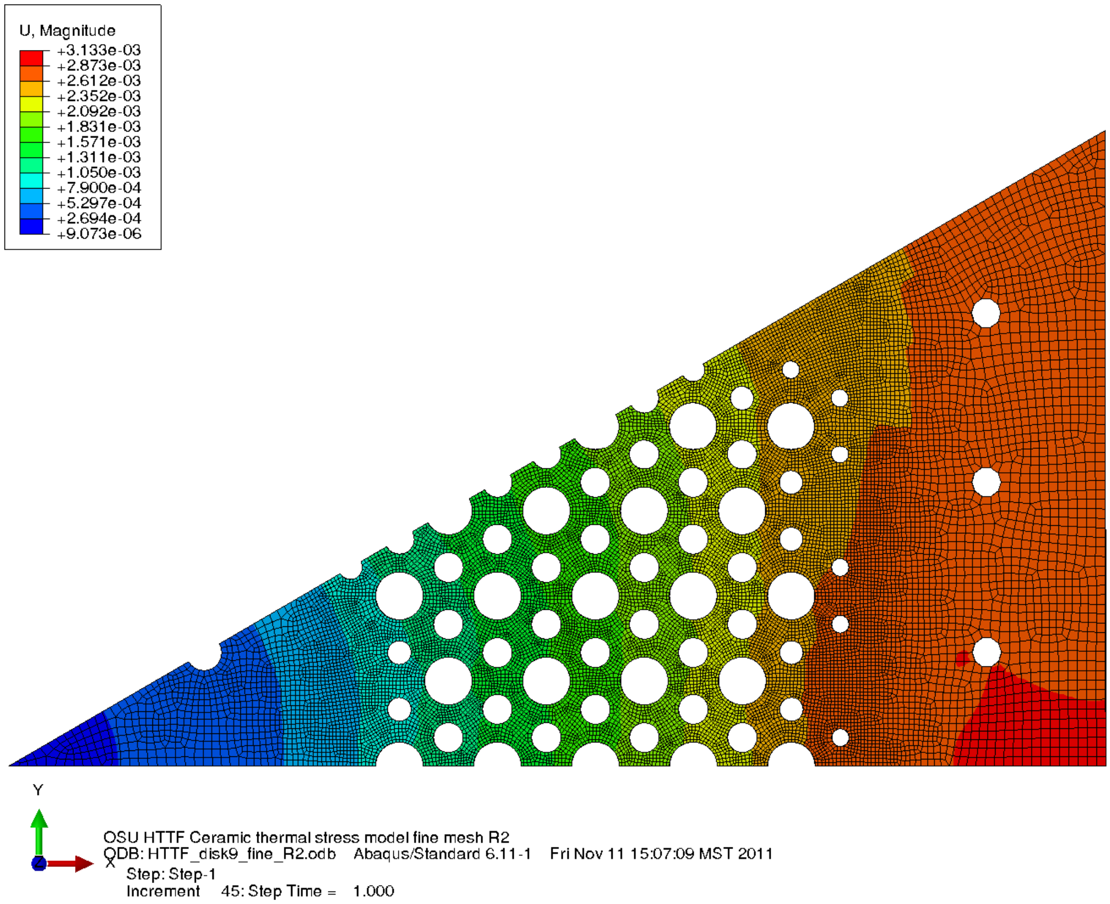


Figure 15. Calculated deformation (m) in the disk.

6. SUMMARY OF HEATUP EXPERIMENTS ON GREENCAST 94F AT OREGON STATE UNIVERSITY [7]

To investigate the thermal gradient effects experience by the ceramic blocks within the HTTF core, a small test sample was created using a prototypical unit cell from the HTTF heated region. The test sample was cast using the technique selected for the final HTTF ceramic components to provide as typical a ceramic block as possible. The test sample was cubic in nature, measuring 7.75 in. on each flat. Figures 16 and 17 below show where multiple channels are oriented in the vertical direction of the block. The small channels are the coolant channels and the larger channels are the heater channels. Numerical modeling conducted before test initiation showed that the greatest thermal stresses resided in the walls of the heater channels. A cartridge heater was placed in one of the heater channels and operated at full power for a period of 2 days. The test ended at this time because of an internal wiring problem within the heater cartridge itself. The heater output was 150% the thermal flux of the prototypical HTTF heating elements. This value was chosen to provide a worst-case heating scenario. Initial scoping calculations demonstrated that the thermal stresses in the heater channel wall were only dependent on the heater heat flux, so the tests were operated in a room that was at ambient temperatures.

The heated channel was inspected for flaws and cracks before and after the heating process. The only observable change in the heated channel was a change in the color of the wall itself, as can be seen in Figures 18 and 19, where the heated channel is the yellowish channel. A container was placed to catch ceramic material that had broken off of the test block during heating. No chips were collected after the heating cycle. During the heating process, thermal expansion was observed in the block and a crack did open up in the block wall at full temperature, but then closed again after the block had cooled down. The major block dimensions were measured before and after heating, with no measureable change.

These tests showed that GreenCast 94F can survive the expected surface heat fluxes of the HTTF ceramic heating system. Thermal expansion was observed during heating, but no permanent changes were observed. Thus, sufficient space surrounding the ceramic materials within the HTTF is necessary, but a component failure is not expected.

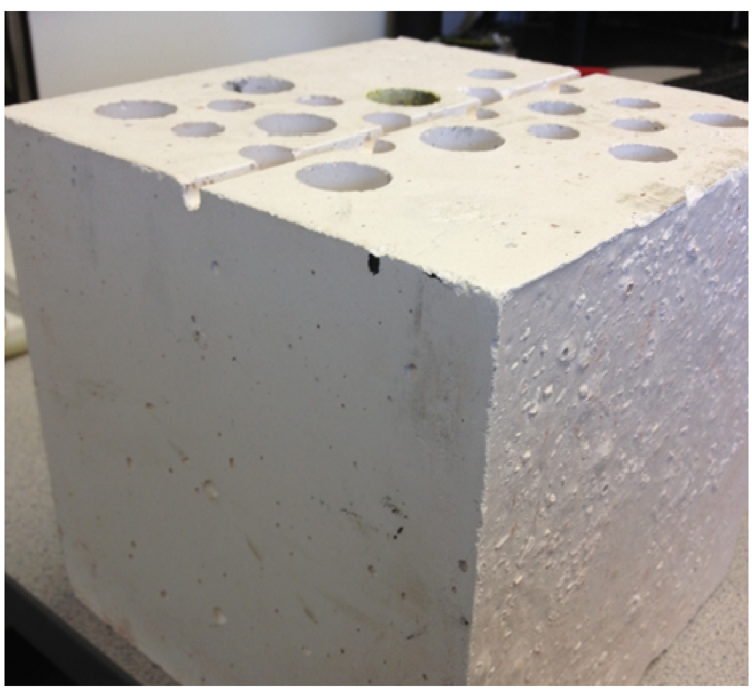


Figure 16. Side view of the test block composed of the GreenCast 94 material.

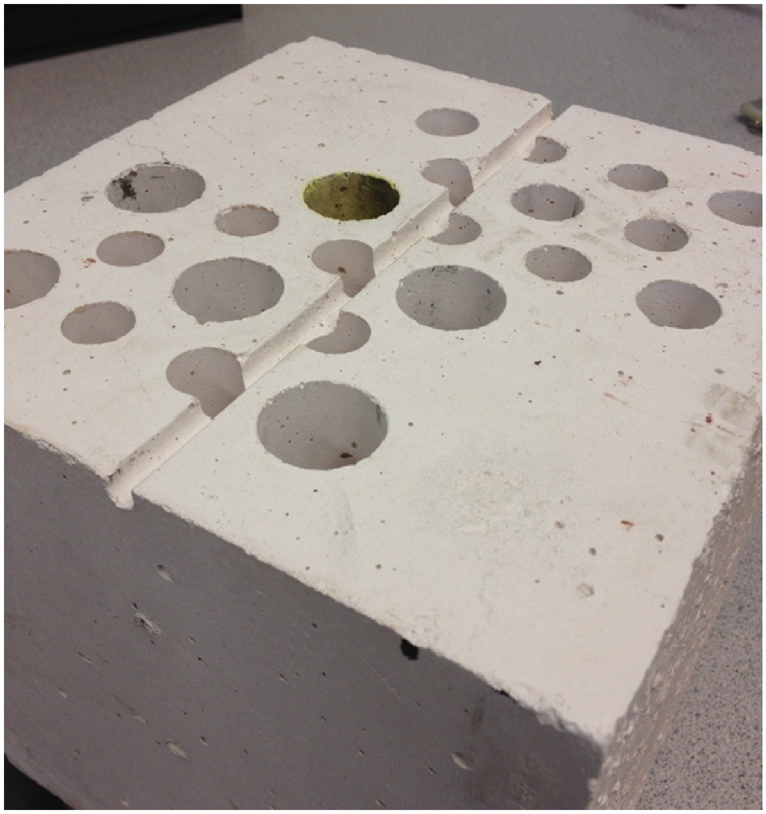


Figure 17. Top view of the test block composed of GreenCast 94.

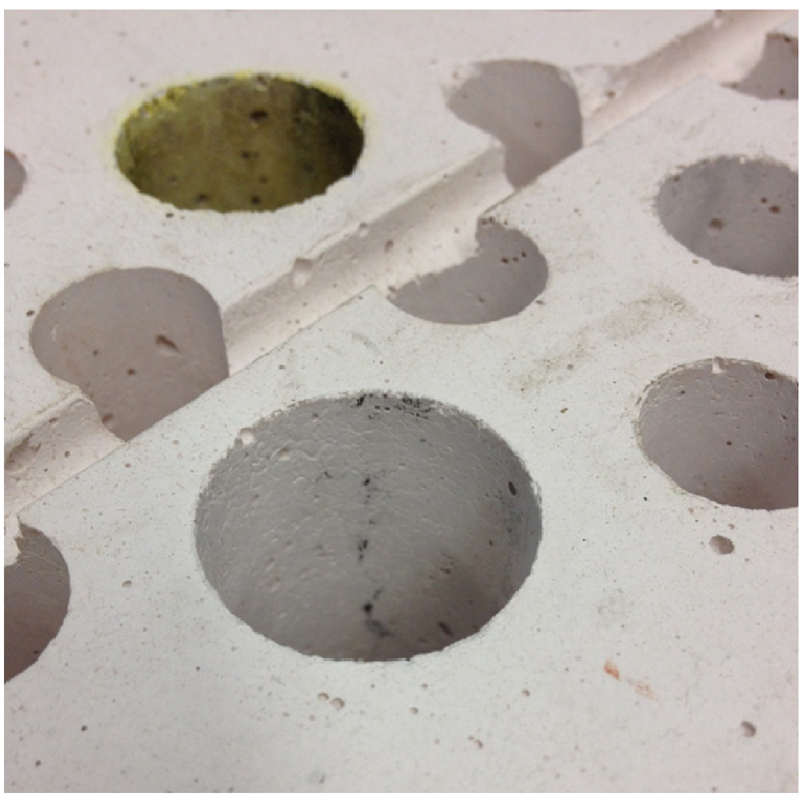


Figure 18. Detail photograph of an unheated channel, center of image, in the test block.

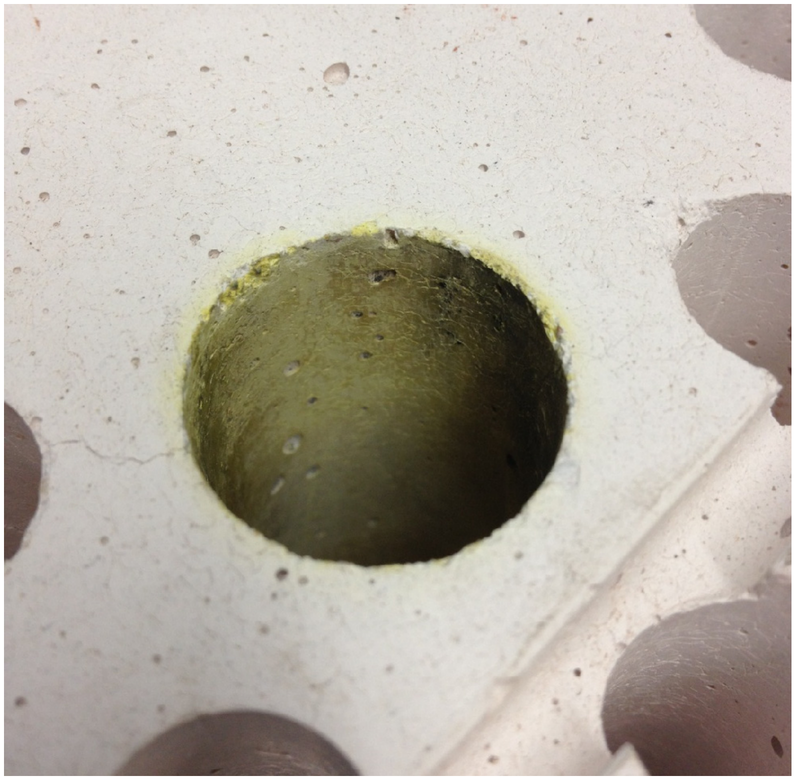


Figure 19. Detail photograph of the heated channel within the test sample.

7. CONCLUSIONS

This thermal stress analysis shows good agreement for mapping temperatures from the STAR-CCM+ model to the Abaqus/Standard model. Using the material properties provided for the GreenCast 94F ceramic, cracking is predicted to occur. However, this modeling does not predict the size or length of the actual cracks. It is quite likely that cracks will be narrow with rough walls which would impede the flow of coolant gases entering the cracks. The first plastic strain observed in the model occurred when the temperature was ramped up to about half of the final temperature, indicating that the temperatures would have to be reduced by about half to prevent the onset of cracking.

However, based on the experimental observations obtained by first heating and then cooling GreenCast 94F ceramic samples at Oregon State University, where only surface cracking was observed, best engineering judgment concluded that cracking detrimental to the experimental program is unlikely.

8. REFERENCES

1. STAR-CCM+, version 5.02.009, 2010, CD-adapco, 60 Broadhollow Road, Melville, NY 11747.
2. RELAP5 Code Development Team, 2005, RELAP5-3D Code Manual, INEEL/EXT-98-00834, Rev. 2.4, June 2005.
3. http://webbook.nist.gov/chemistry/fluid
4. Tung, Y-H., Johnson, R. W. and Sato, H., "Effects of Graphite Surface Roughness on Bypass Flow Computations for an HTGR," paper PVP2011-58035, ASME 2011 Pressure Vessel and Piping (PVP) Division Conference, Baltimore, Maryland, July 17-21, 2011.
5. Abaqus Standard Version 6.11-1, Dassault Systèmes 2011.
6. I-DEAS NX6, Copyright 2008 Siemens Product Lifecycle Management Software Inc.
7. S. Cadell private communication to R. R. Schultz, Oregon State University, 2012

Appendix A

Abbreviated Abaqus/Standard Input File

Appendix A
Abbreviated Abaqus/Standard Input File

The following Abaqus/Standard input file was used to do this modeling.

```
**** file = HTTF_disk9_fine_R2.inp
**%
**%          MODEL FILE: Z:\bdh\Work_Files\NX6\OSU_disks_R2.mf1
**%                 FEM: Fem1
**%
**%               UNITS: SI-Meter (newton)
**%                     ... LENGTH : meter
**%                     ... TIME   : sec
**%                     ... MASS   : kilogram (kg)
**%                     ... FORCE  : newton(N)
**%                     ... TEMPERATURE : deg Celsius
**%
*HEADING
OSU HTTF Ceramic thermal stress model fine mesh R2
**%============================================
**%          MODAL DATA
**%============================================
*NODE, NSET=ALLNODES, SYSTEM=R
       1001, 4.0228380E-01,-3.4694470E-17,-1.8833530E+00
   .....
     270648, 1.1668310E-01, 4.9971660E-02,-2.0814730E+00
*ELEMENT, TYPE=SPRING1, ELSET=SPRING_Z
151, 205578
152, 205593
153, 205653
*ELEMENT, TYPE=C3D8I   , ELSET=SOLID2
     1001,     1001,     1002,     1027,     1026,    17854,    17869,
18244,    18229
   .....
   237790,   226787,   226757,   226142,   226157,   226788,   226758,
226143,   226158
*SOLID SECTION, ELSET=SOLID2, MATERIAL=CERAMIC
*SPRING, ELSET=spring_z
 3
 1.00
*MATERIAL,NAME=CERAMIC
*ELASTIC,TYPE=ISOTROPIC
 4.2265E+10,  0.090
*DENSITY
 2912.6
*EXPANSION,TYPE=ISO,ZERO=293.0
 0.0, 298.1
 6.6667E-06, 373.0
 5.7143E-06, 473.0
 5.4545E-06, 573.0
 6.6667E-06, 673.0
 6.9474E-06, 773.0
 7.1304E-06, 873.0
 7.4074E-06, 973.0
 7.7419E-06, 1073.0
 8.0000E-06, 1173.0
```

```
 8.1026E-06, 1273.0
 8.0930E-06, 1373.0
 8.5106E-06, 1473.0
 8.3951E-06, 1513.0
 7.6078E-06, 1573.0
 5.0909E-06, 1673.0
 6.7797E-06, 1773.0
 1.4603E-05, 1873.0
*CONDUCTIVITY,TYPE=ISO
 5.62022E+00,
*PLASTIC
 17.2e+06, 0.0, 1089.0
 19.3e+06, 0.0, 1644.0
 23.4e+06, 0.0, 1755.0
*NSET,NSET=Y_CONS_ND, GENERATE
      1001,     1025,       24
   .....
    208309,   209028,        1
*NSET,NSET=UPPER_SYM_ND, GENERATE
      5133,     5134,        1
   .....
    209029,   209388,        1
*NSET, NSET=SPRING_ND
  205578,  205593,  205653
*TRANSFORM, NSET=UPPER_SYM_ND
 5.8430160E-01, 3.3734669E-01, 0.0, -0.194767, 0.3373467, 0.0
*INITIAL CONDITIONS, TYPE=TEMPERATURE
 ALLNODES, 298.0
**%
**%==========================================
**%          HISTORY DATA
**%==========================================
**%
**% ====== STEP NUMBER   1 ======
**%
*STEP,INC=1000
*STATIC
 0.005, 1.0, 1.0E-08, 1.0
*BOUNDARY, OP=NEW
 Y_CONS_ND, 2, 2, 0.0
 UPPER_SYM_ND, 2, 2, 0.0
*TEMPERATURE, INPUT=HTTF_disk9_fine_R2_TEMP.inp
*OUTPUT, FIELD
*NODE OUTPUT
 U, NT
*ELEMENT OUTPUT
 S, peeq
*END STEP
```

The temperature input file, HTTF_disk9_fine_R2_TEMP.inp, was edited to have the following format:
node, temperature

```
1001, 1252.65
1002, 1289.55
.....
270647, 746.541
270648, 701.981
```