PS1-1105

Computational thermo-fluid exploratory design analysis for complex ITER first wall/shield components

D.L. Youchison[a], G.O. Natoni[a], M. Narula[b], and A. Ying[b]

Corresponding author: Tel +1-505-845-3138; fax +1-505-845-3130 E-mail: dlyouch@sandia.gov

[a]Sandia National Laboratories, PO Box 5800, Albuquerque, NM 87185 USA

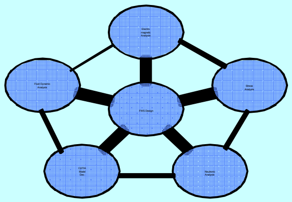
[b]University of California, Los Angeles, CA 90095, USA

SAND2007-6293P

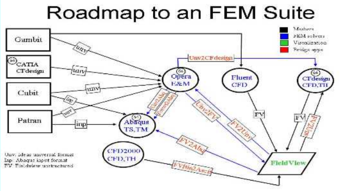
Abstract: Engineers in the ITER US Party Team used several computational fluid dynamics codes to evaluate design concepts for the ITER first wall panels and the neutron shield modules. The CFdesign code enabled them to perform design studies of modules 7 and 13 very efficiently. CFdesign provides a direct interface to the CAD program, CATIA v. 5. The geometry input and meshing are greatly simplified. CFdesign is a finite element code, rather than a finite volume code. Flow experiments and finite volume calculations from SC-Tetra, Fluent and CFD2000 verified the CFdesign results. Several new enhancements allow CFdesign to export temperatures, pressures and convective heat transfer coefficients to other finite element models for further analysis. For example, these loads and boundary conditions directly feed into codes such as ABAQUS to perform stress analysis. We investigated the use of 2 and 4-mm flow driver gaps in the shield modules and the use of 1-mm gaps along the tee-vane in the front water header to obtain a good flow distribution in both the first wall and shield modules for 7 and 13. Plasma heat flux as well as neutron heating derived from MCNP calculations are included in the first wall and shield module analyses. We reveal the non-uniformity of the convective heat transfer coefficient inside complex 3-d geometries exposed to a one-sided heat flux and non-uniform volumetric heating. We obtained temperature and velocity distributions, as well as pressure drop information, for models of nearly exact geometry compared to the CATIA fabrication models. These loads are coupled to thermal stress analysis in ABAQUS. The ITER FW/SM conceptual design may evolve given on-going reviews, but the design process in the USPT has been tested and is ready for the challenge.

Design by Analysis

*Please also see PS1-1108 and PS1-1109



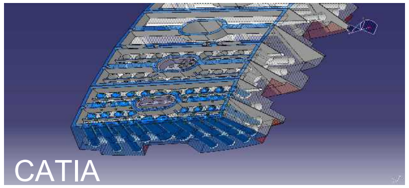
"Design by Analysis" exploits the power of computer-aided design/computer-aided engineering (CAD/CAE) to produce a design that satisfies all the design criteria from multiple disciplines.



Bridging routines can create an FEM suite from a dissimilar set of "best" in-house FE codes.

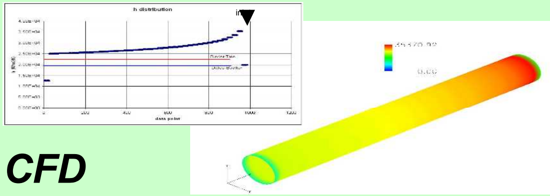
Table 1. Analysis codes and translators

Code name	Discipline	File formats	Sandia translators
CFdesign v9.0, SC/Tetra v4.0, CFD2000 v5.2, Fluent, v6.2	CFD, TH	unv, fld, CATx sat, stl, unv, inp iges, fld sat, fld	Unv2Abq, Unv2FV Abq2FV, Unv2FV FV2Unv FV2Unv
Opera 3-d, v11.1	E&M	unv	Unv2Abq, Unv2FV, Forces2Abq,Unv2CFdesign
Abaqus, v6.2	TS, M	inp	Abq2FV
FieldView, v11.2	visualization	fld	FV2Abq, FV2Unv
Catia, v5.1 R16	CAD	CATpart, CATproduct	

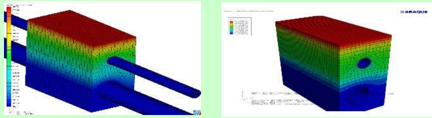


ITER shield module 7 is an example of complex 3-d geometries that require detailed thermofluid and stress analysis.

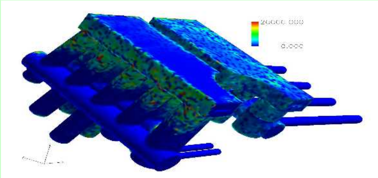
CFD



A comparison of CFdesign computed convective heat transfer coefficients with those from classical correlations for sub-cooled forced convection show good agreement for fully developed conditions (hex mesh).



A comparison between temperature distributions in the solid of two first wall mock-up models shows agreement to within 1 °C (0.4%). A CFD conjugate heat transfer calculation produced the distribution on the left; whereas, Abaqus produced the distribution on the right by application of a 3-d CFD-computed convective heat transfer coefficients to the channel walls.



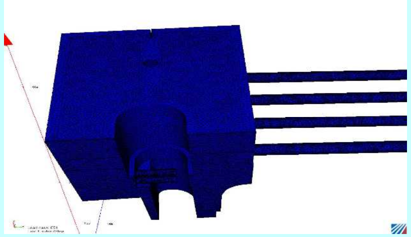
Contour plots of the convective heat transfer coefficient calculated by conjugate heat transfer at the fluid/solid interface reveal enormous variations inside complex 3-d geometries. (Speckling is due to tetrahedral mesh.)

Sandia National Laboratories

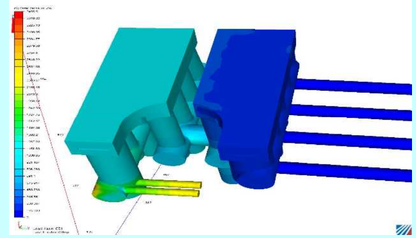
Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

National Nuclear Security Administration

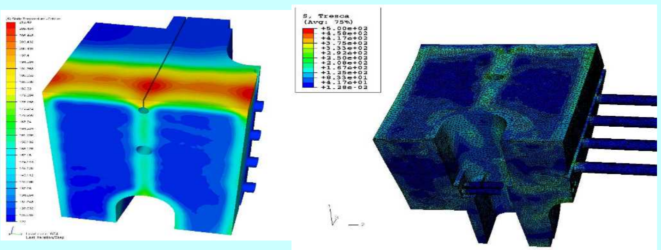
SM7



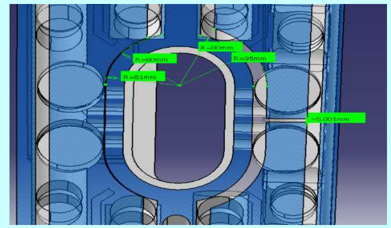
The CFD model for the 1/8 segment of shield module 7 contained over 4 million tetrahedral elements.



The pressure distribution in the SM7 model reveals a total pressure drop of 3 kPa through the quarter. Since two halves are flowed in parallel the total pressure drop across the shield module is 6 kPa.

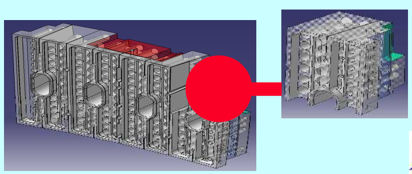


a) The temperature distribution inside the SM7 model shows modest temperatures under steady state ITER operating conditions. b) The maximums in the tresca thermal stress distribution using conservative, purely elastic analysis are well below the 3Sm ITER limit for 316LNG stainless steel.

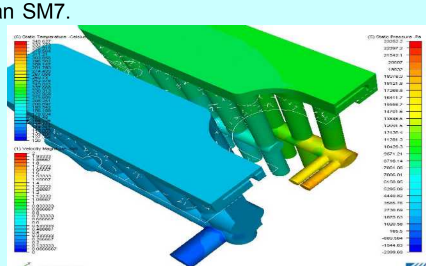


The SM7 tee-vane near the FW leg cutout gap was increased to 15 mm and a 5-mm-slit was cut in the center of the tee-vane to achieve a more uniform flow distribution.

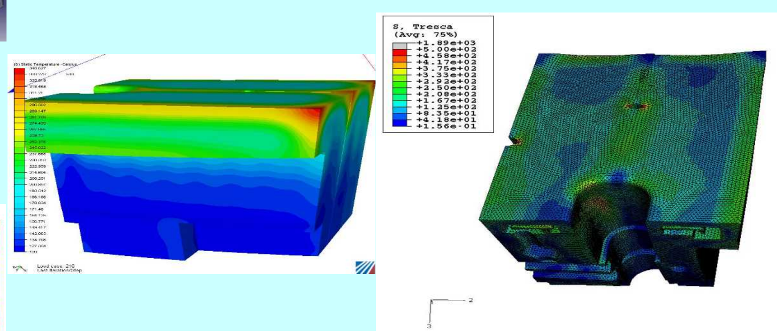
SM13



Cutouts for the water manifolds at the edges of SM13 and its location near the tokamak midplane require the geometry to be different than SM7.

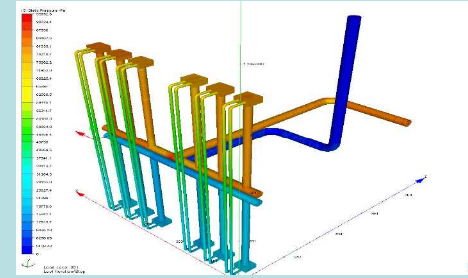


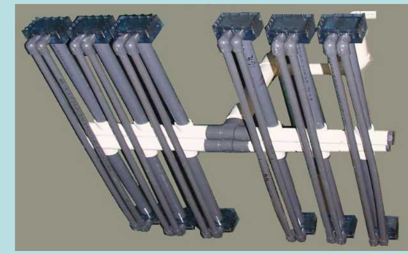
The pressure distribution revealed a total pressure drop of 22 kPa through a 1/8 portion of SM13. The total module pressure drop would be 44 kPa.

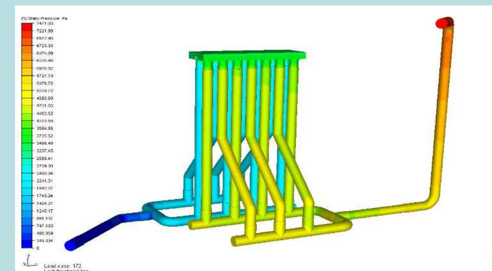


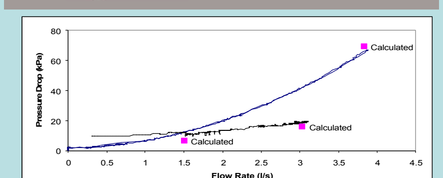
The highest temperatures in SM13 occur at the corners. However, the highest stresses are at the top of the radial holes.

Experimental benchmarks

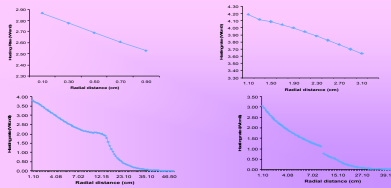




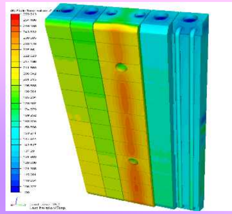


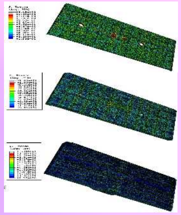


FW7



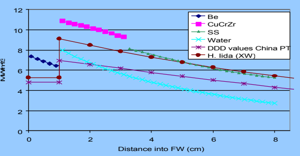
The nuclear heating profiles used for FW7 came from 1-d MCNP-CGM calculations using a radial build of the FW with distinct materials (1d/3d hybrid model). This distribution overestimates power deposition near the surface. A full 3-d analysis is underway.



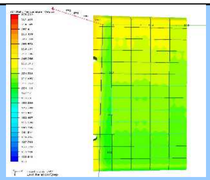


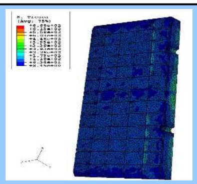
The maximum temperature on the beryllium armor for FW7 is 260 C. A higher temperature stripe corresponds with the access holes. Plasma heating of 0.5 MW/m^2 applied in addition to the nuclear heating.

FW13

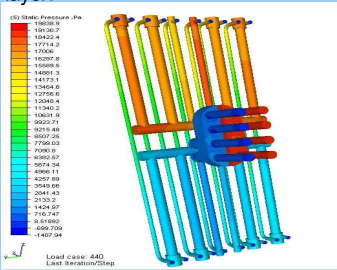


1-d MCNP-GCM calculations that provided the 1-d nuclear heating profile through FW13 are higher than ITER DDD predictions in the CuCrZr layer.





A higher temperature stripe appears along the edge of FW13 where the access holes are located. The maximum beryllium temperature is 216 C. The highest thermal stresses in the beryllium are about 275 MPa.

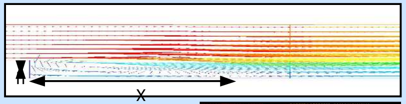


The pressure drop through a FW7 panel is approximately 21 kPa and is due mainly to the 10-mm-dia steel tubes in the CuCrZr layer..

Validation Runs

A Turbulent flow over a backward step

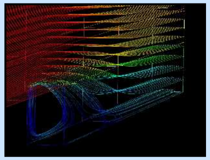
Accurately predict the location of flow reattachment downstream of the step.



Backstep reconnection

Reconnection Length:
X_R = 262.7mm

→ Gives X_R/h = 6.90
→ *(Ref. X_R/h = 6.67)*

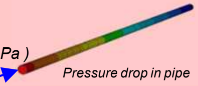


Streamlines over step

B Turbulent pipe flow

Verify the fluid flow and turbulence modeling in a circular pipe. Find the pressure drop from the inlet to 50 meters into the pipe.

Pressure Drop:
→ P = 3.99e5 Pa
→ *(Ref. P = 4.02e5 Pa)*

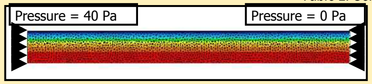


Pressure drop in pipe

Inlet Vel. = 1500 m/s

C Starting Flow in a circular pipe

A uniform pressure gradient is applied through a long pipe to fluid initially at rest. Find the centerline flow velocity as the flow develops.



Axial velocity profile in pipe

Table 2: Centerline axial velocity of fluid at time steps

	Analytical	CFDesign	Error
U (t=0.05)	0.1995	0.1981093	0.70%
U (t=0.1)	0.3851	0.3747758	2.68%
U (t=0.2)	0.6517	0.6467382	0.76%
U (t=0.3)	0.8045	0.8046777	0.02%

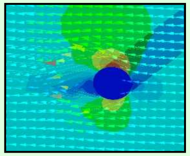
D Drag force on a cylinder

Simulate air flowing past a circular cylinder and find its drag coefficient.

Drag Force = 2.645 lbs.

$$C_d = \frac{DragForce}{0.5\,\rho U^2 Ld}$$

→ Drag Coef. (C_d) = 0.874
→ *(Ref. C_d = 0.91)*

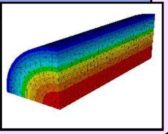


Velocity flow over a cylinder

E Joule heating

Determine the centerline temperature of a steel wire carrying a current and exposed to air.

Centerline Temperature:
→ T_{center} = 420.459 ºF
→ *(Ref. T_{center} = 420 ºF)*

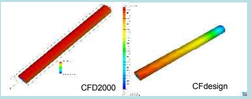


Temperature profile of wire

F Conjugate Heat Transfer

Power = 1000 W

Predict the temperature distribution in a heated copper tube.



Maximum Temperature:
→ *CFD2000: 307.5 ºC*
→ *CFdesign = 307.1 ºC*
→ *SCTetra = 307.1 ºC*

Acknowledgements: We gratefully acknowledge the many contributions of US ITER Party Team members, including M. Ulrickson, T. Tanaka, J. Garde, J. Bullock,,R. Hunt, M. Sawan and P. Wilson.

----- Conclusions -----

The US ITER party team used the "design by analysis" approach to design the ITER first wall/shield modules 7, 12 and 13. Fundamental to this concept is the use of computer-aided engineering tools such as CFdesign for the computational fluid dynamics analyses. CFdesign directly couples to the CATIA v5 CAD software for geometry input. Successive iterations of CFdesign to perform the thermo-fluid analysis followed by immediate design modifications in CATIA lead to rapid optimization of the design. CFdesign was benchmarked against standard CFD problems and results of finite volume codes such as Fluent, CFD-2000 and SC/Tetra. The pressure drop calculations performed by CFdesign were also checked against actual measurements made on full-scale plastic prototypes of module 18.

We developed a suite of translation applets that permit the model mesh and results to be transferred from one FEM package to another with minimal effort. It was demonstrated how the pressure and temperature loads could be calculated in CFdesign and utilized in Abaqus for stress analysis. It was also shown how the 3-d spatial distribution of convective heat transfer can be obtained from conjugate heat transfer calculations and used in Abaqus for fast thermal response/thermal stress studies. The calculated h's were also compared to correlations and highlight the high coefficients that are possible before the thermal boundary layer is fully developed.

Finally, we presented the temperature and pressure distributions obtained for the first wall panels and a ¼ model of the shield modules for modules 7 and 13 under design for ITER. Volumetric neutron heating was included in the CFD models using 1-d distributions obtained from MCNP-CGM calculations. The temperature and pressure distributions obtained from CFD and results of the elastic stress analysis are well within ITER requirements for the original baseline conceptual design.