Proceedings of the 1st Global Power and Propulsion Forum
GPPF 2017
Jan 16-18, 2014, Zurich, Switzerland
www.pps.global

GPPF-2017- 171

HIGH FIDELITY CHT CFD FOR GAS TURBINE HEAT TRANSFER APPLICATIONS

Jose Rodriguez
Siemens Energy
joserodriguez@siemens.com
Orlando, FL, USA

Philipp Cavadini
Siemens Energy
philipp.cavadini@siemens.com
Berlin, Berlin, Germany

Marco Brunelli
Siemens Energy
marco.brunelli@siemens.com
Orlando, FL, USA

Chad Custer
CD-adapco
chad.custer@cd-adapco.com
Northville, MI, USA

Cassandra Carpenter
CD-adapco
cassandra.carpenter@cd-adapco.com
Orlando, FL, USA

ABSTRACT

In an increasingly competitive marketplace for gas turbines, highly efficient and validated computational tools are of paramount importance for reducing the time to market of gas turbine components. Historically, heat transfer has been an area where turbine design has predominantly used one-dimensional correlations, which are based on expensive and very time-consuming rig experiments. With today's available computing power, Conjugate Heat Transfer (CHT) Computational Fluid Dynamics (CFD) simulations allow the characterization of complete sections of gas turbine components, greatly speeding up the design cycle time and reducing the Research and Development (R&D) effort. In this computational paradigm, the quality of the design has to rely on an extensive validation of the CFD tool that becomes an essential part of the design effort. This paper presents a unique CHT CFD process based on Siemens PLM software and the validation of classical cooling flows (turbulators, impingement and pin fins) and full components tested in engines. We show the effect of mesh discretization and turbulence models in producing good engineering results.

INTRODUCTION

The physics associated with the Gas Turbine (GT) is highly complex. Turbulent unsteady flow and heat transfer are two of the fundamental physical phenomena driving gas turbines. Tremendous resources are devoted to designing turbine and compressor blade geometries to improve both aerodynamic performance and control blade temperatures.

CFD has been widely used in the field, primarily for the aerodynamic design of the bladed systems. However, heat transfer analysis has matured more slowly due to the complexity of the modelling required. A traditional approach is to model aerodynamics, solid conduction and cooling features (such as turbulated ducts, impingement and pin fins) separately. These separate models are then loosely coupled to predict an overall system response. Furthermore, empirical relations are used to characterize the cooling features. These methods are rooted in experimental data, which is costly to obtain and significantly slows the pace of development [1-2]. Additionally, these empirical relations are only relevant to a design space similar to the geometry and conditions tested.

Market forces continually press for lower unit cost, lower emissions, higher combined cycle efficiency and faster time to market. Any one of these objectives is a significant challenge. The simultaneous advancement of all objectives requires an evolution in development methods and tools. Using first-principles, physics based models allow a more accurate characterization of the system and the ability to analyse novel concepts outside the scope of empirical methods. Simultaneously solving the aerodynamics, solid conduction and influence of cooling features further increases model reliability and accuracy. Combining these modelling techniques into a streamlined design system shortens design time and allows for efficient design-space exploration.

Significant research is presented each year in the area of CHT CFD analysis. At the 2015 IGTI Turbo Expo alone, numerous papers were presented that focus on numerically analysing impingement cooling [3-7], effusion cooling [8-9], film cooling [5,7,10] and pin fins [11-12]. Additionally, the sensitivity to numerical methods [13], coupling techniques [14-15], and turbulence modelling [15] are active areas of research in turbine heat transfer.

As with all methods, this integrated methodology must be both validated and industrially applicable. This paper validates the CFD solver STAR-CCM+ for fundamental cooling geometries including turbulated ducts, impingement and pin fins. The paper then presents a modelling sensitivity analysis focused on mesh discretization and turbulence modelling. Finally, this integrated method is shown to be viable in a production design environment by analysing a representative gas turbine vane.

The paper first reviews the traditional design process based on correlations, and then presents the complementary process of using CFD and further using CHT CFD in the design process. This is followed by a presentation of the validation with both simple features and complex models including issues related to mesh and turbulence modelling. A summary and conclusions completes the paper.

METHODOLOGY

The different design methods including the methodology for CHT CFD is compared to traditional design methods

Traditional Design Process

By traditional design process, we refer to a process based on correlations obtained with experimental and engine data. In this process a designer, based on requirements and limitations, and using past experience, produces an initial concept and evaluates if it meets the requirements. From this evaluation, a number of iterations evolve until the design meets the requirements. Once the design is mature, it is validated by testing a prototype. Further iterations may be required leading to more prototypes and tests until the design is mature for engine tests and eventually released to production.

Even though the design iterations are fast due to model simplicity (See Figure 1), the validity of the results may be limited to 1D or 2D flow features and may miss important 3D effects. Therefore, testing prototypes is a requirement to mitigate risk before engine tests and further iterations may be needed after engine tests.

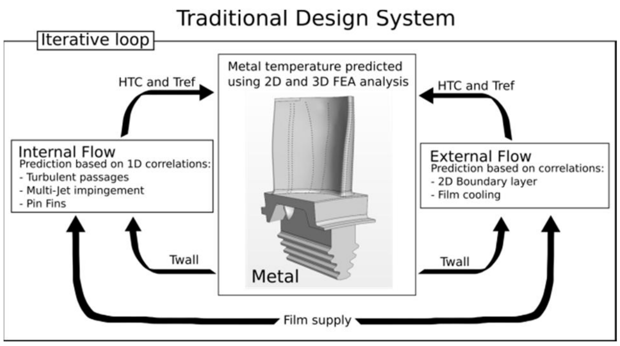

Figure 1. Traditional Design System

CFD Based Design Process

When CFD is introduced early in the design process, it is called CFD Based Design, see **Figure 2.** CFD can replace the last set of iterations with the correlation based design system and the initial prototype testing leading to a faster and less expensive overall maturity of the design. Since an initial concept is required to perform CFD based design, it is only a complement and not a replacement for the traditional design system. In **Figure 2**, the internal and external flows can be solved simultaneously in which case the film cooling is automatically solved.

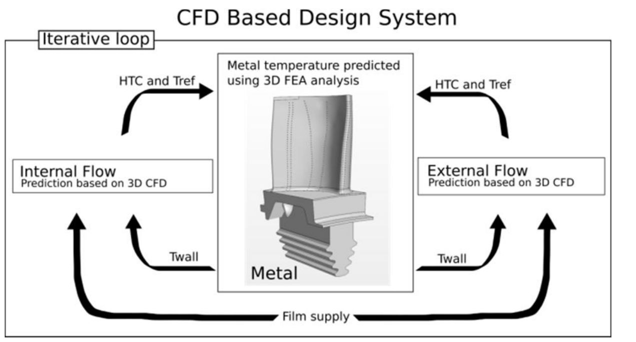

Figure 2. CFD Based Design System

The advantage of this design process is to capture 3D features early in the design iterations therefore helping reduce the number of iterations and prototype testing, in addition, the CFD information can be used to perform specific calibration of the traditional design system for the particular design and increase the validity of the faster design system. A multitude of possible tool combinations is available to the designer, which can then help reduce the design time.

CHT CFD Based Design Process

In recent years, and thanks to the increased computational power, CHT CFD has drastically reduced design iterations and prototype testing, leading to much faster and better designs and enabling advanced designs not possible with the traditional design systems. When CHT CFD is used in the design iterations, it is called CHT CFD based design. Considering **Figure 2**, when the metal and fluid are all solved as part of a single simulation, we call this CHT CFD, see Figure 3. There is no longer need to map boundary conditions because all the equations are solved simultaneously. The main purpose of this paper is to describe the CHT CFD process and its validation.

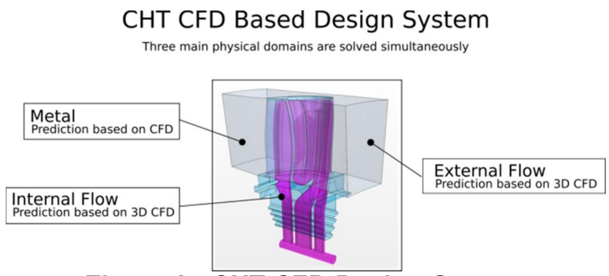

Figure 3. CHT CFD Design System

CAD Preparations For CHT CFD

The CAD software of choice is Siemens PLM NX. The process starts with a hot state CAD model of the metal as shown in Figure 4. This CAD model needs to be de-featured for CFD by removing any manufacturing features that are not required for CHT CFD. Based on the part count a sector CAD model representing all the fluid and solid domains is created as shown in Figure 5, this CAD model is called the "airsolid". This two CAD models are the foundation of the CHT process described in the next section.

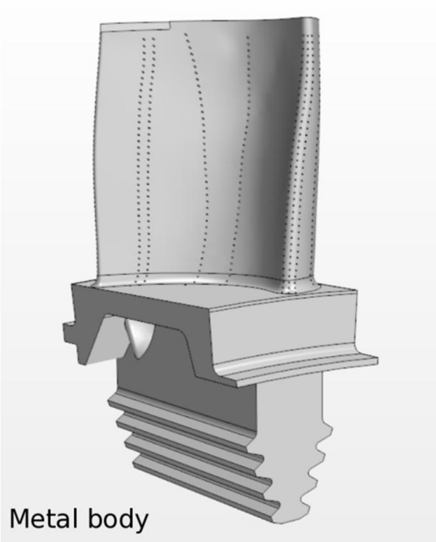

Figure 4. Metal CAD model

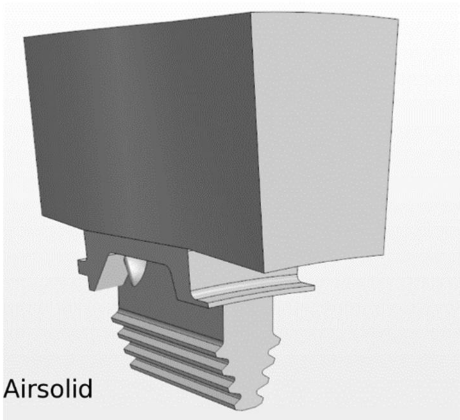

Figure 5. Airsolid

CHT Process and CFD Tool

The CFD tool of choice is Siemens PLM (CD-adapco): STAR-CCM+. The CHT model creation process is executed inside the CFD tool as show in in Figure 6. The Boolean operation between the metal and airsolid, represented by the "+" sign, leads to separate pieces representing the different air cavities within the metal and the connections between them in a manner that the mesh is conformal, and connections with the metal are conformal also. In Figure 6, the purple and grey represent the internal and external airsolids respectively and the light blue the metal. Even though the models in Figure 4 and Figure 5 are simple, the same methodology has been tested in the vane shown in Figure 18 that includes over 700 impingement-cooling jets, several hundred film holes and hundreds of pin fins and other cases with even more complex cooling schemes. The methodology of Figure 6 is easily automated and very robust.

Figure 7 shows the rest of the CHT CFD workflow, including meshing, setting boundary conditions, solving and post-processing.

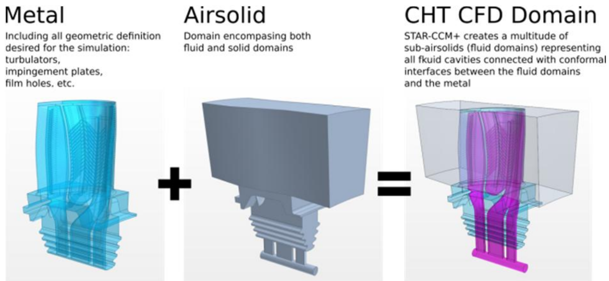

Figure 6. CHT CFD Model Creation Process

It is important to note the while the geometry is created in NX, the final geometry check comes from the surface mesh in STAR-CCM+. The mesh must meet certain quality requirements that increase the likelihood of a successful volume mesh. Any issues with the surface mesh are traced to the CAD models and fixed in NX.

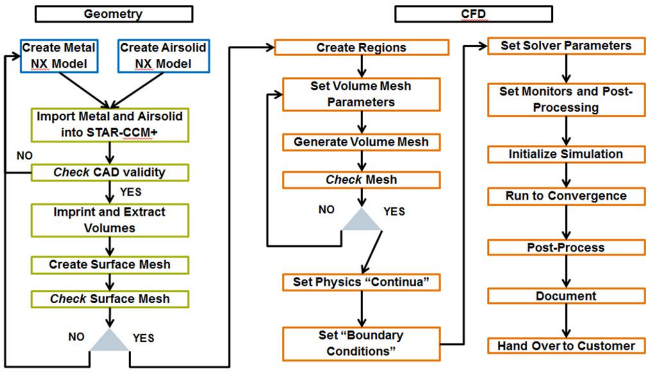

Figure 7. CHT CFD Workflow

RESULTS AND DISCUSSION

One advantage of traditional design systems is that they are based on experimental correlations and engine data and therefore are inherently validated within the accuracy of the correlations and data. On the other hand, CFD is a numerical technique and therefore must be validated. Papers on CFD validation are prevalent in technical conferences. Herein we limit the discussion to the validations done as part of the development of the CHT CFD methodology for the design of GT hot section cooled components. The validation is divided into two types: Feature and Engine. Feature validation is based on academic experiments with simplified geometries and laboratory conditions. These validations are performed in controlled environments and therefore are considered to have less uncertainty than engine validations. Engine validations require real geometries to be tested in an engine at realistic operating conditions. This data typically has more uncertainty due to the complexity of the test environment and the difficulty with instrumentation.

Feature Validation

The feature validation is based on public domain experimental data. Due to differences in laboratory conditions and experimental methods, there is a great variation in experimental results. Due to this variability, the cases are chosen based on perceived quality and completeness of the data in order to minimize modelling assumptions in the "virtual rig" model of the experiment. Due to space limitations, only one example for turbulators, impingement and pin fin are presented.

Turbulated Ducts

The data set for turbulators is 90° ribs from Han [17]. The virtual rig model includes only the hydrodynamic and thermally fully developed section of the rig. Therefore the fully developed interface in STAR-CCM+ is used as shown in Figure 8. The model includes three ribs and the post-processing is done for the center rib.

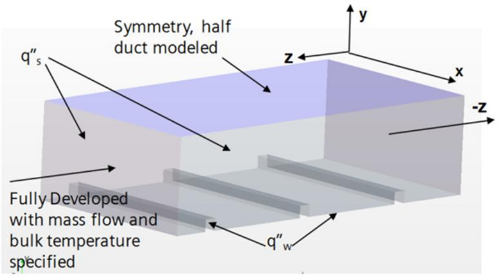

Figure 8. Turbulator virtual rig

Mesh sensitivity shows that the experimental value is approached asymptotically as shown in Figure 9. Note that there is a coarse mesh for which the data is matched perfectly. This illustrates the need for a mesh dependency study. It is possible for under-resolved grids to appear sufficient if analysed in isolation.

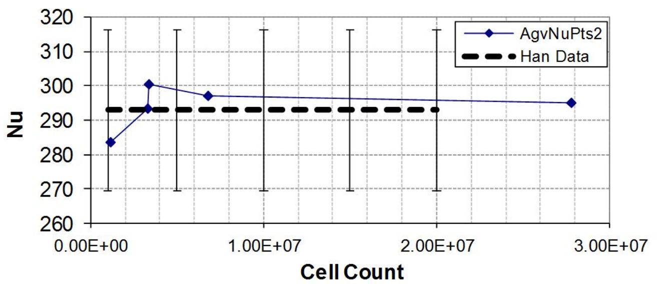

Figure 9. Turbulator Mesh Sensitivity

Figure 10 shows that the V2F matches the heat transfer within experimental uncertainty, while RKE and SST miss by more than 30% using 3 million cells per rib.

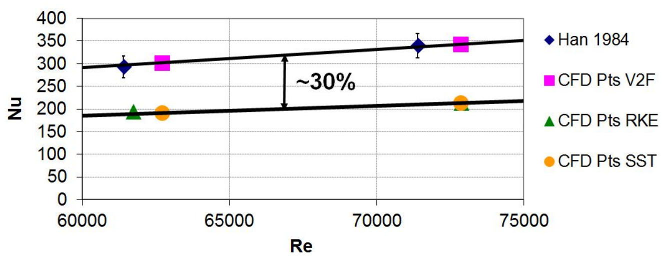

Figure 10. Turbulator Nusselt Number

Impingement

For impingement, two data sets are used, Single jet impingement by Baughn [18] and multi-jet impingement by Florschuetz [19]. The single jet impingement model is shown in Figure 11.

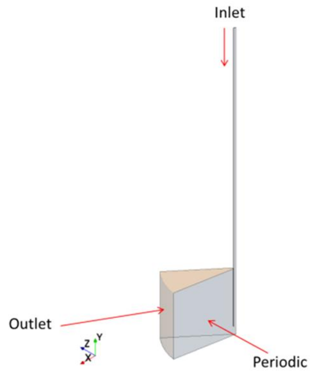

Figure 11. Single Jet Impingement Model

For heat transfer, the best turbulence model is also the V2F with SST as a close second, see Figure 12.

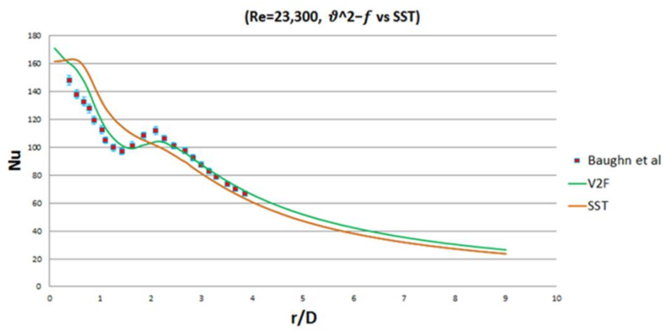

Figure 12. Single Jet Impingement Nusselt Number

The multi-jet impingement model is shown in Figure 13. This parametric model is built using the 3D CAD tool within STAR-CCM+, allowing the automatic simulation of all cases reported by Florschuetz using a simple JAVA code.

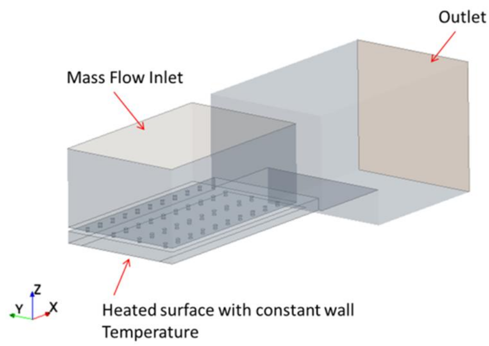

Figure 13. Multi-Jet Impingement Model

Figure 14 shows that both V2F and SST give comparable results; this is the case in general for multi-jet impingement.

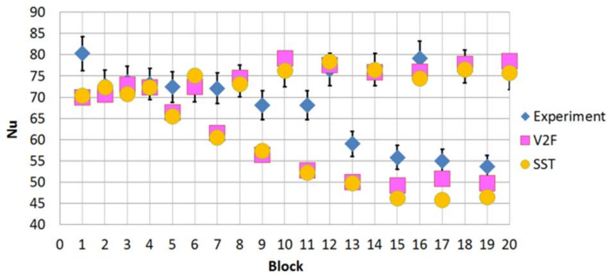

Figure 14. Multi-Jet Impingement Nusslet Number

Pin Fin

The fundamental cooling features known as pin fins were among the more difficult to match with experimental data taken from Ames (19). The inherently complex and unsteady flow phenomena that occurs in a bank of cylindrical pins, includes Von Karman vortex shedding, horse shoe vortices and more. These unsteady flow features influence the time-averaged solution, and limit the accuracy of steady simulation methods.

The computational domain is modelled after the experimental pin bank, Figure 15, with inlet flow conditions and wall heat fluxes of the pins equivalent to the rig test conditions. Resulting steady state simulations did not

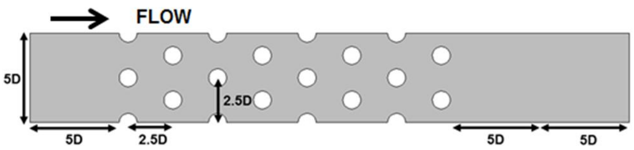

Figure 15. Schematic of the experimental domain for the Ames pin fin bank.

converge to a single steady state solution, but rather oscillated about a mean. The dominant oscillations in the solution field occurred near the wakes behind the pins and resembled unsteady shedding illustrated in Figure 16. With the added computational cost of unsteady simulation, these unsteady flow structures could be fully captured and characterized. However, the purpose of the feature validations is to validate methods for design use where steady simulation is currently preferred. To overcome the numerical oscillations, the solution field was averaged over several hundred iterations to capture a sufficient number of oscillations to ensure a reliable mean. The mean fields of the simulation produced a solution that better resembled a time-averaged solution. This procedure was repeated with all evaluated turbulence models and underwent an extensive mesh sensitivity study.

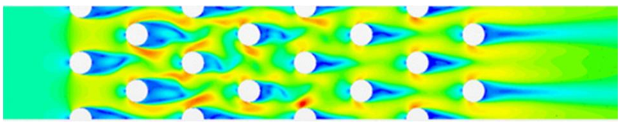

Figure 16. Instantaneous steady state velocity field solution showing chaotic wake behaviour.

The validation also included a URANS investigation with mesh refinement, but no significant advantage was found. A higher fidelity LES investigation did produce better results, however, at this time such studies are considered academic due to the large amount of time and computation resources required. Since we will not be performing full CHT CFD of full components with LES models, the steady state approach is preferred.

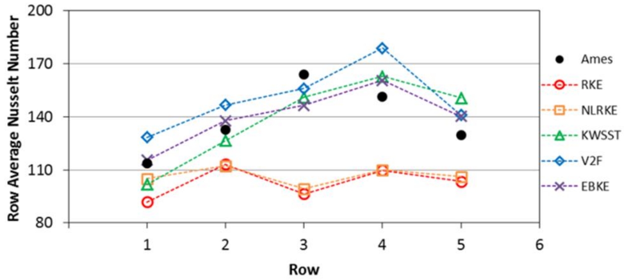

Figure 17. Averaged Nusselt Number results for steady state simulations with various turbulence models.

The averaged results shown in Figure 17 indicate that good trend matching can be achieved with the SST, V2F, and EBKE models. The EBKE model best captured the leading pin result and faired best with the other downstream pins. It is suspected that accurately capturing the leading pin flow phenomena is a requirement for the successful prediction of the downstream flow.

Engine Validation

After the CHT CFD method has been extensively validated with single feature cases, entire components have been investigated. In the following, the comparison of the results predicted by CHT CFD and data obtained from engine tests conducted at the Berlin Test Facility (BTF) for a Vane1 of an H-class engine are presented. Figure 18 shows the computational domain used. The surface of the metal shows the contour plot of the predicted temperature and the streamlines are colored by velocity.

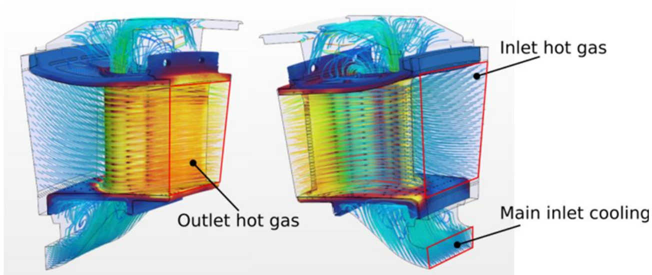

Figure 18. Computational domain of the CHT CFD analysis of a Vane1 of an H-class engine. Metal of the blade is colored by temperature and the streamlines are colored by velocity magnitude.

The metal temperatures are measured with thermal crystals imbedded in the parts. As the boundary conditions differ with the position of the component in the engine, also the predicted results vary. The engine validation included 12 vanes with different technologies for advanced internal cooling instrumented with more than 1500 crystals. Due to space limitations, the quantitative results for a single engine position of the advanced pin fin technology in the trailing edge region are presented. Figure 19 shows the contour plot coloured by temperature of the pressure side of the vane (1), the location of the instrumented thermal crystals (2) and % difference between engine and CHT CFD (3). Fifteen crystals are instrumented on each side. The crystals are grouped in a rectangular grid pattern (3x5). Comparing the results obtained during the engine test with the data predicted by CHT CFD, it shows that the CHT CFD consistently under predicts the temperatures. With a maximum deviation of 8% on the pressure side, the accuracy is excellent.

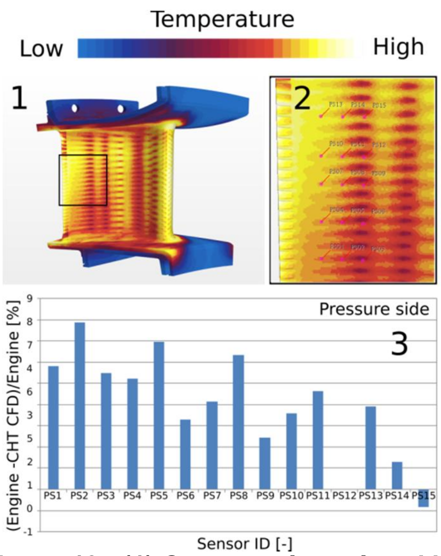

Figure 19. (1) Contour plot colored by temperature of the pressure side of a Vane1. (2) Location of thermal crystals at the tailing edge. (3) Comparison of the temperatures predicted by CHT CFD and engine data.

A similar conclusion is observed on the suction side of the vane as shown in Figure 20. The CHT CFD under predicts the metal temperatures by 7%.

In addition to pin fins, impingement and turbulators have been investigated. The overall comparison between data and CHT CFD is very satisfactory, with maximum deviation no higher than 15%. The methodology is valid for component design.

Mesh and Turbulence Effects

For the methodology development and engine validation, the mesh sensitivity and turbulence model study is based on the results of the feature validation. Unfortunately, the mesh requirements for most of the cases, the mesh size requirement from the feature validation is too large for the full component engine models that include a combination of pin fins, turbulators and mulit-jet impingement. Therefore, the engine validation mesh are coarser then required. Figure 21shows the effect of mesh size and the trend indicates that, for the most part, the results are best for the 185 million cell mesh and worse for the 583 million cell mesh. But all within very acceptable limits -2% to 8% using the RKE turbulence model.

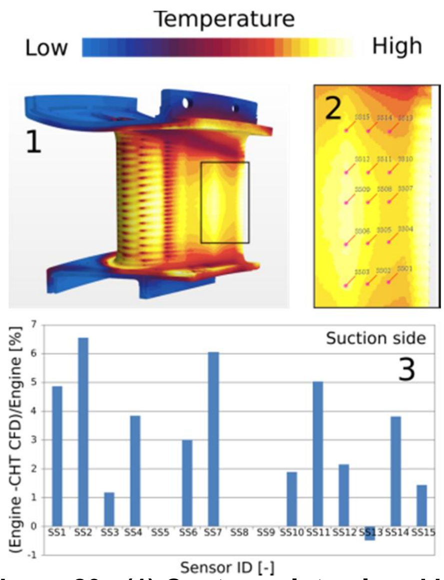

Figure 20. (1) Contour plot colored by temperature of the pressure side of a Vane1. (2) Location of thermal crystals at the tailing edge. (3) Comparison of the temperatures predicted by CHT CFD and engine data.

The effect of turbulence model is shown in Figure 22, using the 80 million cell mesh, the RKE is better than the V2F which is better than the SST. Based on the feature validation, this was unexpected, but since the results are within acceptable range for all cases studied, CHT CFD is considered a validated tool within the limitations of the current validation.

The efforts to reconcile the feature validation and engine validation are on-going by studying larger models, but these are for now only to better understand the results and prepare for when computational resources enable the design with larger models. An effort has been made to use RANS and avoid LES in order for CHT CFD to have a direct day-to-day impact on the component design.

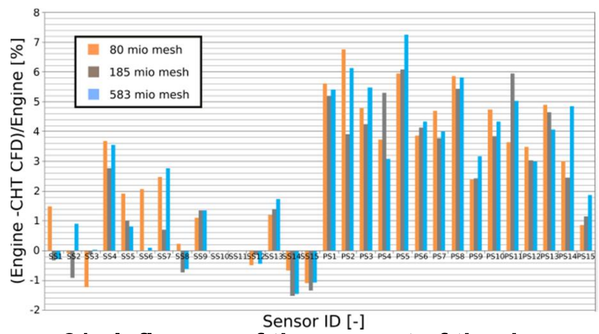

Figure 21. Influence of the amount of the degrees of freedom on the predicted temperatures at the location of the thermal crystals.

CONCLUSIONS

A validated CHT CFD methodology is presented that produces results that are within acceptable limits for gas turbine component design. The validation included both features (turbulators, impingement and pin fins) from laboratory experiments and full components at engine conditions. Even though the mesh for the full component engine validation is too coarse compared to the feature validations and the turbulence model from the engine validation is different than the feature validations, the engine results are within acceptable limits and therefore the methodology can be used to design components.

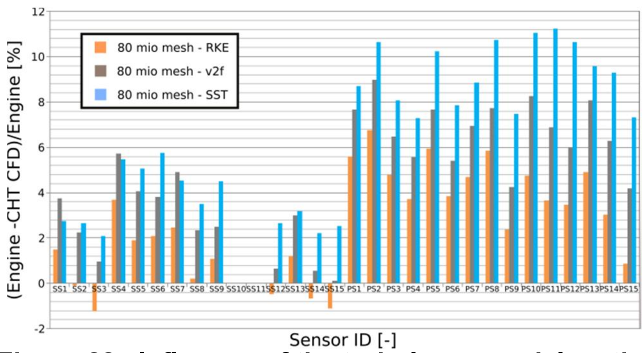

Figure 22. Influence of the turbulence model on the predicted temperatures at the location of the thermal crystals.

NOMENCLATURE

1D	One Dimensional
2D	Two Dimensional
3D	Three Dimensional
BTF	Berlin Test Facility
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
EBKE	Elliptic Blending K-Epsilon
GT	Gas Turbine
HTC	Heat Transfer Coefficient
IGTI	International Gas Turbine Institute
LES	Large Eddy Simulation
Nu	Nusselt Number
NX	Siemens PLM CAD software
PLM	Product Life Cycle Management
PS	Pressure Side
R&D	Research and Development
RKE	Realizable K-Epsilon
SS	Suction Side
SST	Shear Stress Transport
STAR-CCM+	CD-adapco CFD software
Tref	Reference Temperature
Twall	Wall Temperature
URANS	Unsteady Reynolds Averaged Navier-Stokes
V2F	Four Equation K-Epsilon Variant

ACKNOWLEDGMENTS

Many people contributed to the development and validation of this methodology at both CD-adapco and Siemens, we thank both teams for doing a great job.

REFERENCES

[1] Hylton, LD, Mihelc, MS, Turner, ER, Nealy, DA, York, RE, "Analytical and Experimental Evaluation of the Heat Transfer Distribution Over the Surface of Turbine Vanes," NASA-CR-168015, 1983

[2] Hylton, LD, Nirmalan, V, Sultanian, BK, "The Effects of Leading Edge and Downstream Film Cooling on Turbine Vane Heat Transfer," NASA-CR-182133, 1988

[3] Chi, Z, Liu, H, Zang, S, Jiao, G, "Conjugate Heat Transfer Optimization of the Nonuniform Impingement Cooling Structure of a HPT 2nd Stage Vane", Proceedings of the ASME Turbo Expo, 2015, GT2015-42097

[4] El-Jummah, AM, Andrews, GE, Staggs, JEJ, "CHT/CFD Predictions of Impingement Cooling With Four Sided Flow Exit", Proceedings of the ASME Turbo Expo, 2015, GT2015-42256

[5] Li, M, Li, X, Ren, J, Jiang, H, "Overall Cooling Effectiveness Characteristic and Influence Mechanism on an Endwall with Film Cooling and Impingement", Proceedings of the ASME Turbo Expo, 2015, GT2015-43069

[6] Kaçar, EN, Erbay LB, "Numerical Characterization of a Jet Impingement Cooling System Using Coupled Heat Transfer Analysis", Proceedings of the ASME Turbo Expo, 2015, GT2015-43371

[7] Jung, EY, Chung, H, Choi, SM, Woo, T, Cho, HH, "Conjugate Heat Transfer on Full-Coverage Film Cooling With Array Impingement Jets", Proceedings of the ASME Turbo Expo, 2015, GT2015-43810

[8] Andrews, GE, Oguntade, HI, Ingham, DB, Burns, AD, Pourkashanian, M, "The Influence the Number of Holes on Effusion Cooling Effectiveness for an X/D of 4.7", Proceedings of the ASME Turbo Expo, 2015, GT2015-42248

[9] Andrews, GE, Savastrano, W, Ingham, DB, Pranzitelli, A, Biancolini, ME, Pourkashanian, M, "Goal Driven Shape Optimisation for Conjugate Heat Transfer in an Effusion Cooling Plate", Proceedings of the ASME Turbo Expo, 2015, GT2015-42251

[10] Humber, W, Johnson, J, Ni, RH, Clark, J, King, P, "Comparison of Conjugate Heat Transfer Analysis of Flat Plate Film Cooling Arrays Against Experimental Data", Proceedings of the ASME Turbo Expo, 2015, GT2015-42398

[11] Yamane, T, "Numerical Evaluation of Pin Fin Inclination by Conjugate Heat Transfer Simulation with URANS", Proceedings of the ASME Turbo Expo, 2015, GT2015-42869

[12] Li, W, Chi, Z, Kan, R, Ren, J, Jiang, H, "Experimental Investigation of Heat Transfer Dependency on Conjugate and Convective Thermal Boundary Conditions in Pin Fin Channel", Proceedings of the ASME Turbo Expo, 2015, GT2015-42872

[13] Tan, Q, Li, J, Ren, J, Jiang, H, "Second Order Viscous/Heat Fluxes Scheme: Hyperbolic System for RANS Flow and Conjugate Heat Transfer Problem in Gas Turbine", Proceedings of the ASME Turbo Expo, 2015, GT2015-42973

[14] Caloni, S, Shahpar, S, "Investigation into Coupling Techniques for a High Pressure Turbine Blade Tip", Proceedings of the ASME Turbo Expo, 2015, GT2015-43292

[15] Schmidt, M, Starke, C, "Comparison of Steady and Unsteady Coupled Heat-Transfer Simulations of a High-

Pressure Turbine Blade”, Proceedings of the ASME Turbo Expo, 2015, GT2015-43175
[16] Zhang, Z, Xu, H, Wang, J, Li, G, Wang, L, Wu, X, Ma, S, “Evaluation of CFD Predictions Using Different Turbulence Models on a Film Cooled Guide Vane Under Experimental Conditions”, Proceedings of the ASME Turbo Expo, 2015, GT2015-42563
[16] Han, JC., Park, JS., Lei, CK., “Heat transfer and pressure drop in blade cooling channels with turbulence promoters”., NASA-CR-3837, 1984.
[17] Baughn, JW, Shimizu, S., "Heat transfer measurements from a surface with uniform heat flux and an impinging jet," Transactions of The ASME-111, 1989.,pp1096-1111.
[18] Florschuetz, LW, Metzger, DI, Takeuchi, DI, and Berry, RA., "Multiple Jet impingement heat transfer characteristic-experimental investigation of in-line and staggered arrays with crossflow," NASA-CR-3217, 1980.
[19] Ames, FE., Klennert, LA., Nordquist, CA., and Morrow, MJ, “An Experimental and Analytical Investigation of Pin Fin Heat Transfer and Fluid Dynamics for Physics Based Code Development,” 15th ERCOFTAC-SIG15/IAHR Workshop on Refined Turbulence Modeling, 17-18 October 2011, EDF Chatou, France.