2023 INCITE Proposal Submission
Proposal

Title: Toward In-Service Realism: DNS of Gas Turbine Blades with Localized Roughness

Principal Investigator: Richard Sandberg

Organization: University of Melbourne

Date/Time Generated: 6/10/2022 3:02:42 AM

Section 1: PI and Co-PI Information

Question #1

Principal Investigator: *The PI is responsible for the project and managing any resources awarded to the project. If your project has multiple investigators, list the PI in this section and add any Co-PIs in the following section.*

Principal Investigator

First Name

Richard

Last Name

Sandberg

Organization

University of Melbourne

Email

richard.sandberg@unimelb.edu.au

Work Phone

+61 (0) 3 8344 9084

Address Line 1

Department of Mechanical Engineering, Level 4, E407

Address Line 2

(No answer given.)

City

Parkville, Melbourne

State

Victoria, Australia

Zip Code

3010

Question #2

Co-PI (s)

First Name

Ivan

Last Name

Marusic

Organization

University of Melbourne

Email

imarusic@unimelb.edu.au

First Name

Melissa

Last Name

Kozul

Organization

University of Melbourne

Email

kozulm@unimelb.edu.au

First Name

Pawel

Last Name

Przytarski

Organization

University of Melbourne

Email

pawel.przytarski@unimelb.edu.au

First Name

Jake

Last Name

Leggett

Organization

University of Melbourne

Email

jake.leggett@unimelb.edu.au

First Name

Massimiliano

Last Name

Nardini

Organization

University of Melbourne

Email

nardinim@unimelb.edu.au

First Name

Aamir

Last Name

Shabbir

Organization

General Electric Aviation

Email

Aamir.Shabbir@ge.com

First Name

Sriram

Last Name

Shankaran

Organization

General Electric Aviation

Email

shankaran@ge.com

First Name

William

Last Name

Solomon

Organization

General Electric Aviation

Email

William.j.solomon@ge.com

First Name

Paul

Last Name

Vitt

Organization

General Electric Aviation

Email

paul.vitt@ge.com

Question #3

Institutional Contact: *For the PI's institution on the proposal, identify the agent who has the authority to review, negotiate, and sign the user agreement on behalf of that institution. The person who can commit an organization may be someone in the contracts or procurement department, legal, or if a university, the department head or Sponsored Research Office or Grants Department.*

Institutional Contact

Institutional Contact Name

Jason Monty

Institutional Contact Phone

+61 (0) 3 8344 3040

Institutional Contact Email

montyjp@unimelb.edu.au

Section 2: Project Information

Question #1

Select the category that best describes your project.

Research Category

Engineering: Fluids and Turbulence

Question #2

Please provide a project summary in two sentences that can be used to describe the impact of your project to the public (50 words maximum)

Project Summary

This project will improve our understanding of how in-service roughness on turbine blades interacts with complex gas turbine flows by leveraging recent advances in first-principle based turbulence simulation capability. The new knowledge will help improve low fidelity industrial design tools and aid global efforts to reduce fuel use and emissions.

Section 3: Early Career Track

Question #1

Early Career

Starting in the INCITE 2022 year, INCITE is committing 10% of allocatable time to an [Early Career Track](#) in INCITE. The goal of the early career track is to encourage the next generation of high-performance computing researchers. Researchers within 10 years from earning their PhD (after December 31st 2012) may choose to apply. Projects will go through the regular INCITE Computational Readiness and Peer Review process, but the INCITE Management Committee will consider meritorious projects in the Early Career Track separately.

***Who Can Apply**: Researchers less than 10 years out from their PhD that need LCF-level capabilities* *to advance their overall research plan and who have not been a previous INCITE PI.*

How to Apply:

In the regular application process, there will be a check-box to self-identify as early career.

- *The required CV should make eligibility clear.*
- *If awarded, how will this allocation fit into your overall research plan for the next 5 years?*

Projects will go through the regular INCITE review process. The INCITE Program is targeting at least 10% of allocatable time. When selecting the INCITE Career Track, PIs are not restricted to just competing in that track.

- *What is the Early Career Track?*
  - *The INCITE Program created the Early Career Track to encourage researchers establishing their research careers. INCITE will award at least 10% of allocatable time to meritorious projects.*
- *Will this increase my chances of receiving an award?*
  - *Potentially, this could increase chances of an award. Projects must still be deemed scientifically meritorious through the review process INCITE uses each year.*
- *What do I need to do to be considered on the Early Career Track?*
  - *In the application process, select 'Yes' at 'If you are within 10 years of your PhD, would you like to be considered in the Early Career Track?' You will need to write a paragraph about how the INCITE proposal fits into your 5-year research and career goals.*
- *What review criteria will be used for the Early Career Track?*
  - *The same criteria for computational readiness and scientific merit will be applied to projects in the Early Career Track as will be applied to projects in the traditional track. The different will be manifest in awards decisions by the INCITE management committee.*

Early Career Track

If you are within 10 years of your PhD, would you like to be considered in the Early Career Track? Choosing this does not reduce your chances of receiving an award.

No

If 'yes', what year was your PhD? If 'no' enter N/A

N/A

If 'yes', how will this allocation fit into your overall research plan for the next 5 years? If 'no' enter N/A.

N/A

Section 4: INCITE Allocation Request & Other Project Funding/Computing Resources

Question #1

OLCF Summit (IBM / AC922) Resource Request - 2023

Node Hours

416000

Storage (TB)

130

Off-Line Storage (TB)

130

Question #2

OLCF Frontier (Cray Shasta) Resource Request – 2023

Node Hours

960000

Storage (TB)

100

Off-Line Storage (TB)

100

Question #3

OLCF Frontier (Cray Shasta) Resource Request – 2024

Question #4

OLCF Frontier (Cray Shasta) Resource Request – 2025

Question #5

ALCF Theta (Cray XC40) Resource Request - 2023

Question #6

ALCF Polaris Resource Request - 2023

Question #7

ALCF Polaris Resource Request - 2024

Question #8

ALCF Polaris Resource Request - 2025

Question #9

ALCF Aurora (Intel X^e) Resource Request – 2023

Question #10

ALCF Aurora (Intel X^e) Resource Request – 2024

Question #11

ALCF Aurora (Intel X^e) Resource Request – 2025

Question #12

List any funding this project receives from other funding agencies.

Funding Sources

Question #13

List any other high-performance computing allocations being received in support of this project.

Other High Performance Computing Resource Allocations

Section 5: Project Narrative and Supplemental Materials

Question #1

Using the templates provided here, please follow the [INCITE Proposal Preparation Instructions]to prepare your proposal. Elements needed include (1) Project Executive Summary, (2) Project Narrative, (3) Personnel Justification and Management Plan, (4) Milestone Table, (5) Publications Resulting from prior INCITE Awards (if appropriate), and (6) Biographical Sketches for the PI and all co-PI's. Concatenate all materials into a single PDF file. Prior to submission, it is strongly recommended that proposers review their proposals to ensure they comply with the proposal preparation instructions.

Concatenate all materials below into a single PDF file.

1. **Project Executive Summary (One Page Max)**
2. **Project Narrative (15 Pages Max)**
3. **Personnel Justification and Management Plan (1 Page Max)**
4. **Milestone Table**
5. **Publications resulting from prior INCITE Awards (if appropriate)**
6. **Biographical Sketches for the PI and all co-PI's.**

INCITE2023_Combined_Final.pdf
The attachment is on the following page.

PROJECT EXECUTIVE SUMMARY

Title: Toward In-Service Realism: DNS of Gas Turbine Blades with Localized Roughness

PI and Co-PI(s): R D Sandberg (PI), I Marusic, M Kozul, P Przytarski, M Nardini, J Leggett, A Shabbir, S Shankaran, W Solomon, P Vitt

Applying Institution/Organization: The University of Melbourne

Number of Processor Hours Requested: SUMMIT: 416,000 (1/2 year), FRONTIER: 960,000 (1/2 year)

Amount of Storage Requested: 230TB

Executive Summary:

Surface roughness affects fluid flow in many situations, from the atmospheric boundary layer always encountering rough surfaces, to many engineering applications where initially smooth surfaces may become rough due to fouling or erosion. An engineering application where roughness effects can significantly reduce efficiency (by up to 10% in several components) is the gas turbine (GT). Given that in 2019 in the USA alone GTs produced 38.4% of all power generated (1.58×10^9 MWh) and burned 18×10^9 barrels of jet fuel, roughness effects have a tremendous impact on cost and emissions. Therefore, any engine performance improvements realized through better understanding and prediction of roughness effects can have a fuel-spend advantage of order billion-$, together with a significant CO_2 emission benefit, and would also increase the viability of costlier, more sustainably sourced fuels. Additionally, better prediction of how roughness, developing during GT operation, affects performance and heat transfer can substantially increase operability, durability and life of GTs.

Understanding and predicting roughness effects in GT flows is particularly challenging, for two reasons. Firstly, realistic roughness topologies from in-service engines can be very intricate and cannot easily be characterized by simple concepts, such as a sandgrain roughness. Secondly, the flow fields in GTs are much more complex than in previously studied applications such as ship hulls or pipelines. They include severe pressure gradients, separation and reattachment, wake and shock-boundary layer interactions, laminar-turbulent transition and relaminarization, and deterministic unsteadiness, to name a few. How localized and topologically complex surface roughness interacts with and affects those phenomena is far from fully understood and no models exist that are generally applicable and accurate.

The current proposal will address the problem of realistic roughness effects in the context of GT flows at realistic engine conditions using first-principle based simulations. For the first time, a systematic high-fidelity simulation study will be conducted of realistic heterogenous roughness obtained from in-service engines, in the complex flow fields that occur in high-pressure turbines. The generated data will shed light on the detailed fundamental physical mechanisms associated with the effects of roughness in boundary layers exposed to curvature, pressure gradients and external perturbations in form of freestream turbulence and incoming wakes. Furthermore, it will help evaluate and develop lower order models readily applicable to GT designs and other applications. Due to the micro-scale of the surface roughness, incredibly fine grids are required to resolve their interaction with the flow in detail. Therefore, the planned simulations are extremely computationally intensive and only possible by applying a highly efficient CFD solver, here HiPSTAR, developed by the PI's group, on a leadership computer facility. The code has been extensively optimized for the GPU architecture of Summit and currently is being optimized for Frontier's architecture.

The University of Melbourne (AU) team has an extensive track record in high-fidelity simulation of turbulent flows in GTs. At the same time, the team at GE has been active in the field of CFD quality and predictability improvement in GT flows and has been leveraging data from hi-fi numerical approaches to GT designs. The proposing team is therefore uniquely qualified to conduct and interpret simulations that will advance the understanding of the effects of localized surface roughness in complex flows, with relevance to GT environments. This project is expected to identify opportunities to increase GT turbine performance and extend hot-gas-path durability. The study will provide information on how different roughness distributions modify the complex gas turbine flow phenomena and thus affect performance and blade life. This knowledge can help design degradation-tolerant blades that maintain higher efficiencies in operation and therefore provide economic and environmental benefits over the entire life cycle.

PROJECT NARRATIVE

1 SIGNIFICANCE OF RESEARCH

1.1 Background

Surface roughness affects fluid flow in many applications, from the atmospheric boundary layer, which always involves rough surfaces, to many engineering applications, where initially smooth surfaces may become rough due to fouling or erosion. An engineering application where roughness effects can have a particularly large impact on cost and emissions is the gas turbine (GT). Previous experimental studies have shown that roughness-effects can reduce efficiency of both compressors and turbines of order 10% [1]. Given the vast world installed base, any GT efficiency increase realized through better understanding and prediction of roughness effects has significant potential to reduce fuel burn and environmental impact. The General Electric GT installed base alone burns \$150B/yr in oil and gas. Every percentage point increase in combined cycle efficiency would reduce GE GT fuel costs by \$1.5B/yr and reduce CO_2 emissions per MW by 1.5%. Crucially, lower fuel consumption would also incentivize uptake of more expensive sustainable fuels. Improved prediction of how roughness that develops during GT operation affects performance and heat transfer can also substantially increase operability, durability and life of GTs.

In GTs, the high-pressure turbine (HPT) experiences high levels of unsteadiness and turbulence, as well as the highest temperatures, pressures, and velocities. These extreme operating conditions result in high Reynolds numbers and transonic Mach numbers, generating highly complex flow and thermal field features that include a multitude of laminar-turbulent transition mechanisms, pressure gradients and shock waves interacting with wakes and boundary layers, to name a few. To complicate matters further, surface roughness resulting from in-service degradation or from novel additive manufacturing techniques can considerably affect the blade boundary layers in the turbine and, consequently, their aerodynamic efficiencies. A loss in turbine efficiency clearly impacts the overall efficiency of the GT in converting the energy stored in the fuel (whether current fossil-based, sustainable fuels or hydrogen) into thrust or power. Additionally, these roughness effects can have even more severe consequences: they can significantly promote heat transfer from the gas to the metal components and thus ultimately determine the life of components. A detailed understanding of the complex flow phenomena in the hot-gas-path, and their interactions with surface roughness, is therefore of paramount importance for both users and developers of GTs.

Yet, accurate prediction of the aerothermal behavior in the presence of realistic roughness is a formidable challenge, due to the complex flow physics and the small scales and complex topology of the surface roughness. To date, the majority of research on turbomachinery performance has been carried out by costly physical tests. Moreover, physical testing often can only provide integral parameters of components, e.g. the heat transfer [2], rather than providing sufficient detail to dissect the range of physical phenomena impacting the performance of a design. For that reason, and with continued increase of computing power, Computational Fluid Dynamics (CFD) has become a key GT design tool. However, its full potential is far from being fully realized due to challenges in simulation and modeling of turbulence. In practice, design iterations are accomplished by modeling all scales of turbulence with the well-known Reynolds Averaged Navier–Stokes (RANS) and Unsteady RANS (URANS). While fast, these methods suffer from accuracy issues in the presence of complex mixing processes [3, 4] or roughness effects [5] which limits the true benefit CFD can have on technology development. Inherently unstable processes present the greatest challenge for such numerical tools: for example the accurate prediction of wake-blade interactions or unsteady shock-boundary layer interactions [6]. Nevertheless, highly accurate solutions are essential: for example, a 2% error in the predicted metal temperature is estimated to halve the effective blade life [7].

High-fidelity simulations, particularly Direct Numerical Simulation (DNS), have been shown to be very accurate in simulating turbomachinery flows [8, 9, 10, 11, 6]. By resolving all turbulent scales in the unsteady flow field of GT components, such simulations are able to provide insight into complex flow physics including the impact of curvature and pressure gradient on boundary layers, and the two-way interaction between turbulence, wakes, and surface-roughness [12] on boundary layer development. The principal challenge for DNS of rough-wall flows is the tremendous computational resource required. In order to accurately capture all relevant physical mechanisms, from the unsteadiness at large scale to the smallest turbulence scales and geometric details of surface roughness, high-quality grids often needing more than 10^9 points are necessary. At the same time, more than 10^7 time steps (or increments/realizations) must be computed, such that the resulting simulations will have 10^{16} degrees of freedom. Computations at this scale are only possible with the resources provided by leadership class computing facilities that now allow DNS of realistic rough-wall GT flows at engine relevant conditions.

1.2 State of the art

As surface roughness can significantly affect the performance of GTs and other engineering equipment, the roughness effects on boundary layer flows have been extensively studied, mostly via experiments [1, 13]. While there has been some consensus that wall-roughness results in extra wall friction and enhanced heat transfer compared to smooth wall flows, the detailed underlying mechanisms for such roughness effects are far from fully understood. Furthermore, although empirical models have been proposed to account for the effects of wall roughness [14], predictive models that can accurately represent roughness effects on boundary layer losses and surface heat transfer for a range of configurations and operating conditions do not exist.

To further understand the detailed mechanisms of roughness effects, high-fidelity simulations, mostly DNS, of rough-wall boundary layers have been conducted in recent years. Most of the existing studies, however, have focused on canonical flows such as channel flows [15], pipe flows [16] and flat-plate boundary layers [17], with the roughness elements often limited to regular configurations like cubes [18] or egg-cartons [19]. The University of Melbourne group, in particular co-PI Marusic and collaborator Jelly (who led the simulation campaign on Summit in 2021), have used experiments and high-fidelity simulations to significantly contribute to the understanding and modeling of turbulent boundary layers [20] and roughness effects in canonical configurations [12, 16]. Specifically, Jelly's recent work has focused on using a surface generation algorithm to systematically identify the key topographical parameters that govern the fluid dynamic properties of rough-wall turbulent flows. Jelly and Marusic have also recently performed the very first DNS of pulsatile rough-wall turbulent pipe flow, which highlighted the impact of flow unsteadiness upon the near-wall region, and, in particular, the impact of unsteady forcing upon flow reversal and boundary-layer separation below the roughness crests.

Considering the inherent unsteadiness of turbomachinery flows, as well as the critical role that flow separation, reversal and reattachment play in determining the aerothermal performance of turbines (and GTs in general), we expect similar insights to be gained from the proposed numerical campaign, as well as a unique opportunity to identify the key differences (or similarities) between the unsteady rough-wall flow dynamics in canonical and complex flows configurations. In fact, as summarized by the recent review papers [11, 6], high-fidelity simulations, particularly DNS, of GT components with realistic geometry and engine relevant conditions are still very limited, especially for HPT and high-pressure compressors (HPC) operating at high Reynolds numbers. This is because DNS of GT flows at engine-relevant conditions with realistic roughness are extremely challenging, as *1)* the Reynolds number in the high-pressure parts of GTs are very high, i.e. $Re = 5 \times 10^5 \sim 2 \times 10^6$ in the high-pressure turbine (HPT), incurring vast computational cost; *2)* extra mesh refinement is needed to accurately resolve the flow disturbances caused by the micro-scale roughness elements; and *3)* realistic roughness from in-service GTs can be extremely complex, thus introduces extra uncertainties [21, 22]. Because of these difficulties, high-fidelity simulations of roughness effects in turbine flows so far have mostly been limited to low-pressure turbines at relatively low Reynolds numbers and selected roughness configurations [23], or canonical configurations as mentioned above, with the only exception being the proposers' INCITE 2021 activity that simulated HPTs including roughness that had constant height around the entire blade [24].

The very first DNS of a linear HPT nozzle was performed by the PI and colleagues in 2016 [10], and this is part of a long-term effort by the GE - University of Melbourne collaboration to address the key challenges in turbomachinery flows via high-fidelity simulation. The proposers to date have successfully performed a series of high-fidelity simulations of GT flows with a wide range of configurations, including various compressors, low-pressure turbines (LPT) and HPT with increasingly complex configurations [4, 8, 9, 25, 26, 27]. Based on the generated data, our understanding of these problems has been significantly improved. In two subsequent INCITE grants (TUR105), highly-resolved LES of an isolated HPT nozzle midspan section [10, 26, 28, 29] were performed, and the effects of varying Reynolds number ($Re = 5.7 \times 10^5 \sim 1.1 \times 10^6$), inlet turbulent intensity ($Tu = 5\% \sim 20\% U_{in}$) and turbulent length scale ($LS = 5\% \sim 20\% C_{ax}$), and exit Mach numbers have been investigated. Highly resolved LES investigations of HPT stages have been performed with another INCITE(2020) grant by the proposing team. In the stage calculations, the inflow turbulence was again varied and the interaction of the freestream turbulence and the wakes coming off the stator with the rotor smooth-wall boundary layers studied [30]. The Hi-Fi nature of the HPT simulations allowed a remarkable match with laboratory data in terms of isentropic Mach number (indicative of loading) (see Fig. 1 a).

Given the high impact the above previous studies have had, this proposal now intends to bring together the different expertise and extend the previous GT simulations to consider the roughness effects in HPT blades and stages operating at engine-relevant conditions and with realistic roughness topologies. The proposed simulations will distinguish themselves from previous studies in two principal aspects:

- It will be the first attempt of DNS of a realistically-roughened HPT blade. Our simulation campaign as part of INCITE2021 considered different levels of **homogeneous roughness** (i.e. same mean amplitude over the entire blade). Even considering homogeneous roughness, the type of typical small-scale roughness that exists on an in-service HPT can be extremely complex, and the characterization of the surface roughness is thus non-trivial [22]. For that reason, prior to this INCITE2021 campaign, all high-fidelity studies on roughness effects addressed canonical flow cases, or LPT at relatively low Reynolds numbers. In a final simulation as part of INCITE2021, we considered a blade that had a **varying roughness height over its chord-length**, which was informed by our project partners from GE. In particular, the leading edge of in-service blades will tend to have a larger amplitude roughness due to its direct exposure to the incoming combustion products. We found this varying-height case to radically alter the transition location of the turbulent flow over the blade, and therefore the heat transfer (Fig. 2b). Given the significant changes we observed, **this preliminary case motivates a systematic investigation of how the location of the rough-to-smooth transition affects the complex fluid dynamics behavior**. The boundary layer flows in HPTs are subject to a number of phenomena, including surface curvature, especially near the blade leading edge, relaminarization caused by the strong flow acceleration upstream of the suction peak, and transition to turbulence in the adverse pressure gradient part of the suction-side boundary layer. During the INCITE2021 campaign, different levels, and different distributions, of surface roughness was found to significantly interact with these complex physical mechanisms, which in turn will have significant impact on the turbine efficiency and heat transfer.

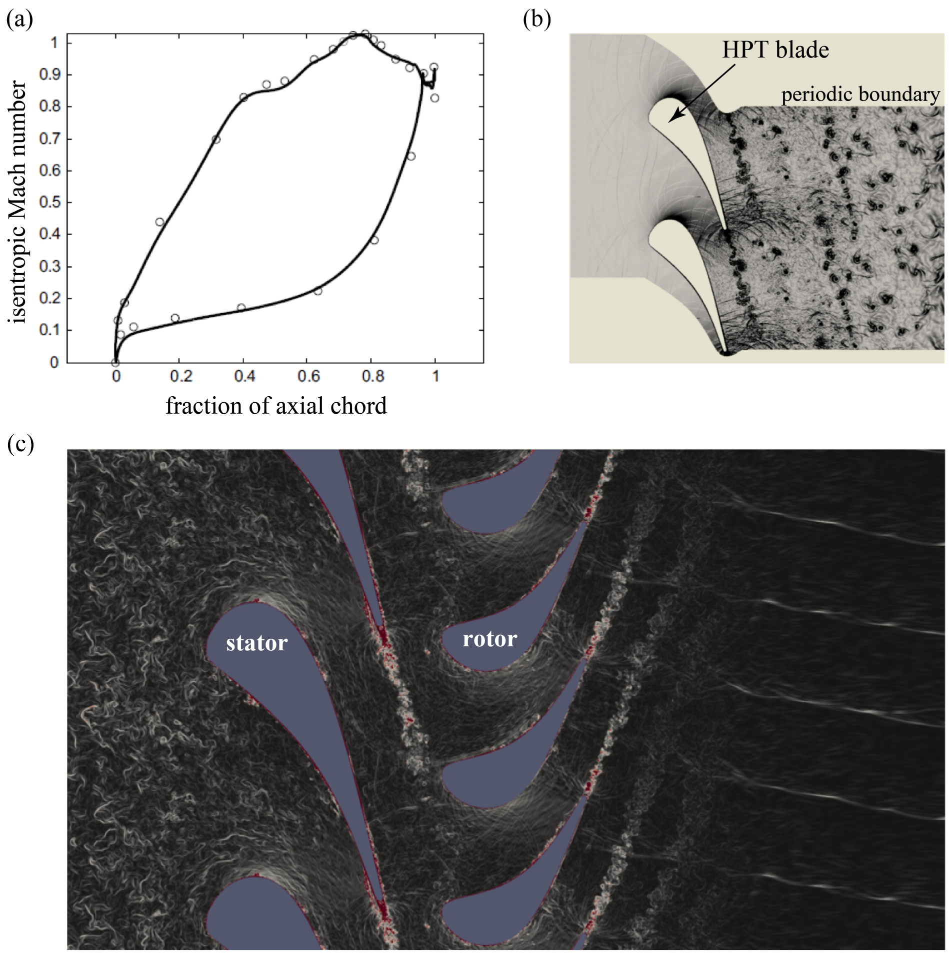

Figure 1. (a) Plot showing high agreement between measurements and DNS predictions of isentropic Mach number (indicative of loading) for an HPT [26]; (b) Numerical Schlieren showing shocks and flow structures in the flow field around a smooth HPT blade with laminar inflow; (c) smooth HPT stage configuration with incoming turbulence [30], instantaneous contours of the density gradient magnitude shown on a spanwise plane-cut.

- We will conduct the first DNS of a realistically-roughened HPT blade in a stage configuration, to study the effect of incident flow disturbances, i.e. both freestream turbulence and discrete wakes. This is important given that turbines are never quiescent and free stream forcing often results in (unsteady) transition and changes in boundary layer behavior over the blade. This will allow the significance of roughness to be accurately compared with other flow effects for the first time in a numerical configuration pushing toward realism.

1.3 Anticipated Impact of the proposed simulation

With computational resources available through the INCITE program, it is finally possible to perform highly detailed numerical studies of roughness effects on GT components under realistic conditions. The DNS data for the HPT with roughened surfaces indicative of in-service patterns will be analyzed with advanced post-processing methodologies and serve as input to model development techniques our groups have developed

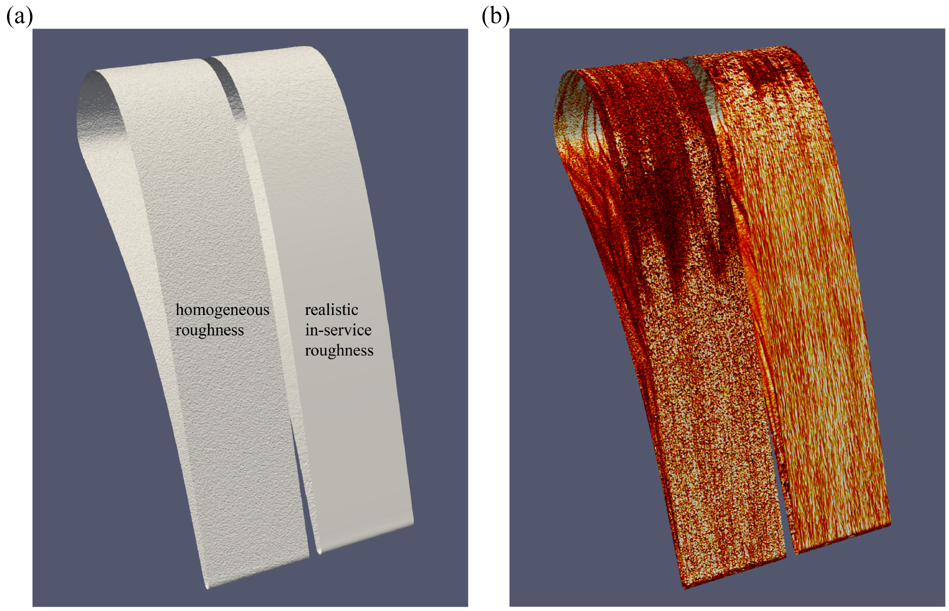

Figure 2. Simulations completed as part of INCITE2021: (a) HPT blade with homogeneous roughness (k/C = 0.003) alongside an HPT blade with a realistically-roughened surface (i.e. leading edge significantly more roughened), a preliminary case motivating the current work; (b) heat transfer coefficient over the surface of the two blades.

so far, and thus the data will support the research and design communities in several ways.

Firstly, the DNS data can accurately resolve multi-scale physics and detailed unsteadiness of the flow, thus improve our understanding on the complex physical mechanisms including flow separation, relaminarization, laminar-turbulent transition, shock-boundary layer interactions, and particularly their interactions with the varying roughness over the length of the blade. These phenomena, despite some having previously been studied individually in more canonical cases, have to date not been studied in combination, and will most certainly reveal even more intriguing physical mechanisms. For example, the laminar-turbulent transition has been extensively studied due to its importance in both the fundamental and applied realms of fluid mechanics, and it has been reported that the transitional mechanisms can be affected by various factors, such as free stream turbulence [31, 32, 33], stream-wise pressure-gradients [34, 35], and curvature of the leading edge [36]. However, the existing knowledge, which mainly comes from studies on relatively canonical cases like flat-plate boundary layer transition, is dominated by single selected factors, and does not necessarily guarantee a thorough understanding on the transition mechanisms in HPT boundary layers with surface roughness. In fact, the transition in HPT suction-side boundary layers is usually affected by a combination of different mechanisms, including a blunt leading edge with high-curvature, strong pressure-gradient varying along the blade, and violent free stream turbulence (up to $Tu = 20\%U_{in}$) and incoming wakes which has not been considered previously in the context of wall roughness. Therefore, to thoroughly understand the laminar-turbulent transition in HPTs with surface roughness, detailed analysis based on the proposed DNS is required. This understanding will then also be applicable to other configurations in which boundary layers form in high-curvature environments producing high pressure gradients and substantial flow acceleration, such as aircraft and other aerospace equipment. Moreover, the role of large-scale turbulent structures, so-called 'superstructures' [37], is relatively well understood in the context of canonical smooth- and rough-wall flows, but their role (or even their existence) remains largely unknown in the context of turbomachinery flows and could finally be clarified with the proposed simulations.

Secondly, the DNS data will be used to validate and develop low-order models and design tools, to investigate and improve the accuracy of RANS solvers/models. Although different modeling strategies have been applied to account for roughness effects on boundary layer development and transition [38, 39], these roughness models are mainly based on empirical correlations from experimental data and the predictive accuracy is often not satisfactory. The novel HPT cases proposed in the present study will be the first gold-standard data sets on wall roughness encompassing turbomachinery flow regimes. Thus, based on direct analysis of the high-fidelity data, we will be able to not only quantify the error from design tools, but also develop improved roughness models via machine learning methods. In recent years, data-driven turbulence model development has attracted substantial interest, and encouraging results have been reported with trained models applied to different cases [40], including the modeling of the effective roughness heights [41]. Nevertheless, challenges still exist in developing novel turbulence closures for practical applications, and the availability of suitable training data seems to be one of the most prominent issues. To be more specific, the success of machine-learnt models so far is limited to relatively canonical cases, in which the complex flow physics in practical applications like GTs are not well represented. In fact, in order to develop models showing advanced predictive accuracy in engineering applications like HPT with surface roughness, data from the proposed high-fidelity simulations of corresponding configurations are needed.

Finally, the understanding on roughness effects drawn from the present study could guide future designs of industrial turbomachines and help increase life and maintain performance through better deterioration models. The surface roughness of GT components becomes increasingly prominent at the first signs of degradation, for example by cracking of thermal barrier coatings. Therefore, GT operators will likely see a gradual drop in performance efficiency. In severe cases, excessive temperatures may also occur in the turbine due to enhanced heat transfer, causing a reduction in life. The ability to accurately predict turbine losses arising from surface degradation will help operators better forecast performance and fuel costs. The proposed simulations could also shed some light into whether erosion-tolerant blades could be designed that exploit the roughness effects on transition. Finally, the physical insights gained could also address the key question as to how smooth blades need to be manufactured such that detrimental roughness effects can be negated. This is highly relevant in the context of additively manufactured (AM) components, as the cost and time required for additional postprocessing of surfaces to remove AM-roughness might outweigh the benefits the new manufacturing technologies deliver. Therefore, a detailed understanding of the roughness effects on the performance of GT components, which could be drawn from the proposed simulations, is highly relevant to the design community.

In conclusion, with the availability of a leadership computer, this proposal aims to perform first-of-a-kind DNS of HPT and realistic patterns of surface roughness and at engine relevant conditions. The data obtained from the simulations will not only extend our fundamental understanding of laminar-turbulent transition, relaminarization, and interactions between shock and wakes with rough-wall boundary layers, but also help evaluate and develop lower-order models readily applicable to GT designs. The findings from the proposed simulations, including both physical insights and models developed with the help of the data, will be published in archival journals and presented at the leading conferences on fundamental fluid mechanics and turbomachinery. General Electric's involvement in this project will also ensure industrial impact of the new knowledge and models, and, in the long term, design improvements derived from the proposed simulations will produce economic and environmental benefits. Finally, we will make our best efforts to provide the community open access to the data, so that the larger engineering and scientific communities can benefit, creating even further impact.

2 RESEARCH OBJECTIVES AND MILESTONES

2.1 Objectives

Our previous work has demonstrated the sensitivity of GT flows at high Reynolds number to roughened surfaces. The world-first attempt at high-fidelity numerical study of the roughness present on all real blades was part of our INCITE2021 project. That effort principally studied homogeneous roughness over the chord-length of the HPT blade [24]. **The proposed research strides toward in-service realism by building on our previous activities in high-fidelity simulation of GT flows with a systematic study of localized roughness as it truly occurs on HPT blades that are actively in use**. This proposal therefore builds on previous simulations at realistic turbine operating conditions with homogeneous wall roughness. As well as the numerical framework, this effort is bolstered by the scientific experience of members of the assembled team in both numerical and experimental study of the effect of roughness on turbulent, wall-bounded flows. The simulations will be conducted on the first stage HPT blade due to the availability of reference simulations with both smooth-walls and homogeneous roughness (i.e. same mean roughness height over entire chord-length) completed in previous INCITE projects (TUR105, ENG112). Subsequently, our request on Frontier will be for an HPT with localized roughness in a stage configuration, to study include the effect of incident flow disturbances including intermittent wakes. This high-fidelity study of turbulent flow facing localized roughness (i.e. higher-amplitude roughness over the leading edge) over a curved wall with pressure gradients will be a valuable addition to the literature, as roughness has been previously studied in simplified, canonical setups. Additionally, from an immediate industrial viewpoint, these simulations represent a major step forward in accurate representation of HPT (and more broadly, GT) flows, where small differences in predicted performance have huge implications due to the vast world installed base.

The proposed simulations of localized, realistic roughness distributions over HPT blades will be undertaken over a 12-month period. The specific objectives of the research efforts can be summarized as:

- **(a) Variation of localized roughness on HPT.**
  - ∗ DNS of HPT with various patterns of localized (i.e. hetereogeneous) roughened surfaces will be performed. Our INCITE2021 simulations were the world-first high-fidelity simulations of HPT at engine-relevant flow conditions resolving the micro-geometry of a roughened blade. Naturally these first simulations considered homogeneous (same mean amplitude) roughness, but a final simulation of that campaign considered localized roughness, namely that the leading edge was significantly rougher than the rest of the blade (Fig. 2a). This localized distribution was informed by our project collaborators at GE as being representative of a blade that has experienced erosion. **We found a drastic change in heat transfer due to this more realistic roughness distribution, which warrants further investigation as detailed in this proposal (Fig. 2b).** Therefore different patterns of localized roughness which might be expected over the lifecycle of the HPT will be investigated.
  - ∗ By analyzing and comparing the detailed flow features and performance parameters from these simulations, we will be able to assess current roughness models, that are largely based on over-simplified configurations [22]. More importantly, the unique data sets will allow development of low-order models with improved predictive accuracy and guidelines for what salient features of surface roughness should be included in future studies and design tools. Advanced post-processing (e.g. entropy loss method [42]; loss analysis based on the Mechanical Work Potential [43, 44]) will allow us to quantify the local loss generation due to roughness, on both the suction or pressure sides.
- **(b) HPT stage configurations**
  - ∗ Following the simulations in set (a), we will undertake the first DNS of realistically-roughened HPT blades in a stage configuration (Fig. 2b). This will generate a realistic unsteady incoming flow to the roughened HPT rotor (the stator will remain smooth), including impinging wakes which are known

to have a large effect on flow transition over the chord-length of the HPT rotor. Simulating the stator as well as the rotor is necessary to correctly capture the asymmetric, wake-laden flow incoming to the roughened rotor, as well as potential effects between the blades. **Synthetic representations of this inflow to the rotor will likely not correctly capture these combined effects, therefore the full stage configuration is required.** Since a stage simulation is much more costly, a smaller number of limiting roughness cases from set (a) will be selected for this configuration (i.e. those roughness distributions that significantly changed the transition location of the flow). It is critical to investigate roughness in a realistic stage configuration to understand the combined effect of the incoming flow disturbances and the roughness on the flow characteristics over the chord-length as we push toward truly realistic simulations of the turbine.

* Given the importance of the impinging wakes from the stator on the roughened HPT rotor, phase-lock averaged statistics will be collected to characterize the evolution of the stator wakes in the rotor passages. It has been shown that the stretching and deformation of the stator wakes is dominated by the mean flow shear, and their interactions with the rotor blades can significantly intensify the heat transfer on the suction side [30].

As a reward for the tremendous computational effort required, the model-free DNS approach offers the highest possible accuracy. The simulations will be carried out with the code HiPSTAR, which has been thoroughly tested and evaluated for GT flows with smooth walls [10, 8, 9, 26] and GT flows with surface roughness [23, 24]. The proposed approach will lead to the generation of gold-standard data sets that can be employed by the broader research community to improve its understanding of unsteady flow physics in high-pressure gradient, high-curvature boundary layers and GTs with localized roughness. Additionally the data can be fed into recently developed machine learning strategies that can build novel design tools with improved predictive accuracy that account for realistic roughness distributions as found on GT blades.

2.2 Overview of Simulations

In Q1 of the proposed research, we plan to expand on our previous simulations of a roughened HPT blade to more comprehensively investigate the effect of having realistic, localized roughness over the chord-length. The simulations will be run at an engine-relevant Reynolds number of 590,000 and a Mach number of 0.9 (based on exit-conditions), making them the first DNS of realistic, localized roughness effects on an HPT blade. The proposed cases for the HPT alone for the first half-year are outlined as cases T1 to T4 in Table 1 (a). The plan of action is to begin with an HPT that has a leading edge (LE) minimally-eroded, as it would presumably be early in its life-cycle, and progressively increase the extent of the high LE erosion in subsequent simulations. This set of cases may also include a case with skewed roughness, representative of that resulting from cracking of the thermal barrier coating used on HPT. This will elucidate at which point flow characteristics, namely transition and heat transfer, are significantly altered by this increasing localized roughness of the in-service blade. These textured surfaces will be incorporated via an immersed boundary method [45] implemented in the code HiPSTAR. Note that the roughness will be applied to significant parts of the blade surfaces, resulting in varying mean roughness height k^+ (scaled with the local viscous length scale) values on suction and pressure surfaces, and interaction with differing flow phenomena depending on the state of the flow at that chord location. This will therefore provide an opportunity for more fundamental physical insight into turbulent flows over roughened, curved surfaces, and subject to strong pressure gradients. We will leverage our experience running HPT at engine-relevant conditions to set-up and run these simulations, with synthetic inflow turbulence mimicking incoming disturbances from an upstream combustion chamber [24]. The DNS will provide a comprehensive data set to investigate the parameterization of roughness and its representative effects in real engineering problems in comparison to canonical cases.

Table 1. Details of the overall computational effort

Case	Re	Case	System	Node hours
(a) Variation of localized roughness on HPT				
T1	590k	Minimal extent of LE roughness	Summit	48,000
T2	590k	Intermediate extent of LE roughness 1	Summit	48,000
T3	590k	Intermediate extent of LE roughness 2	Summit	48,000
T4	590k	Maximal extent of LE roughness	Summit	48,000
(b) HPT stage configurations				
S1	590k	rotor 1 rough	Summit	224,000
S2	590k	stator & both rotors rough, distribution 1	Frontier	480,000
S3	590k	stator & both rotors rough, distribution 2	Frontier	480,000
Total				**1,376,000**

Over Q2-Q4 we will undertake simulations of the HPT blade with localized roughness in a stage configuration (cases S1-S3). The precise roughness distribution, importantly the extent of high amplitude, leading edge roughness (from in-service erosion), or skewed roughness (representing cracking of the thermal barrier coating used on HPT) will be informed by the investigation in set (a). Roughness distributions of interest will be selected for the stage configurations, i.e. cases with significant changes in the location of transition to turbulent flow over the HPT, and therefore significantly altered aerothermal performance. Roughness effects in the presence of incident wakes has never been investigated in detail with DNS before, but is key to understanding the role roughness plays in real turbines. S1 will only have one rotor with a roughened surface, with the stator and the second rotor smooth (each stage configuration features one stator and two rotors, see Fig. 3). This will not only constrain the cost of this case to be run on Summit, but will allow direct comparison of a smooth and rough rotor seeing the same incoming flow from the stator, allowing isolation of the roughness effects during post-processing. For stage configurations S2 and S3, the roughness distributions chosen will again be informed by the investigation of set (a) (cases T1-T4), however in S2 and S3, all three of the stator and two rotors will be roughened. The two rotors could additionally have different roughness distributions, allowing us to study (between S2 and S3) a total of four localized roughness distributions in the stage configuration. In summary, cases S1-S3 will permit a more accurate assessment of the roughened blades' performance in a stage configuration, where they will be faced with the true incoming flow of in-service turbine blades.

2.3 Post-processing of Data

The true value of the huge amounts of data that will be generated with the proposed simulations can only be unlocked with cutting-edge post-processing. Our post-processing can be broadly classified into two categories:

a) Examination of the flow behavior through visualization, modal decomposition (e.g. POD [46]) and novel entropy loss analysis [42]. Those can improve understanding of the physical mechanisms and the loss generation processes in the rough-wall GT components, particularly in the presence of background turbulence, transition and discrete periodic disturbances, like wakes and shocks. Together with the mechanical work potential approach [47], this will allow identifying the contributions of both suction and pressure side roughness to overall performance changes; and

b) Machine-learning approaches to develop lower order models with improved accuracy that can effectively model the effect of localized roughness. Importantly, in RANS calculations, of the type used in industry, roughness effects have so far been characterized by only selected parameters, e.g. equivalent sandgrain height [39], which is known to be over-simplified and lacks accuracy for practical problems [48]. The novel machine learning methods developed by the Melbourne group [49, 50], previously shown to produce CFD-ready symbolic models with improved accuracy from INCITE-generated data sets, will be applied to the new data sets. Models that can more accurately characterize the roughness effects in realistic GT flows will be developed and then validated in RANS calculations.

3 COMPUTATIONAL READINESS

3.1 Use of Resources Requested

The required grid points and iteration steps shown in Table 2 are estimated based on DNS and LES conducted on Summit and other supercomputers during previous allocations, including INCITE projects TUR105, ENG112 and ALCC project TUR123. Our previous LES of rough-wall HPT blades [24] of a single HPT vane at $Re = 5.9 \times 10^5$ required 3.4 million in-plane grid points. The blade boundary layers were well-resolved with averaged near-wall grid spacings of $\langle \Delta s^+ \rangle \lesssim 1.0$ (tangential), $\langle \Delta n^+ \rangle \lesssim 1.0$ (wall-normal) and $\langle \Delta z^+ \rangle \lesssim 10.0$ (spanwise), normalised with the local viscous length scale. These spanwise-then-time-averaged grid spacings were evaluated at the mean roughness height, coinciding with the coordinates of the smooth HPT blade. The increased resolution requirements around the blade are offset by savings away from the blade enabled by the overset mesh technique now used in HiPSTAR [51] that was not available for the previous DNS study [26]. Compared to our INCITE2021 study, in order to push toward a DNS resolution (i.e. no subgrid-scale turbulence model), we will increase the spanwise resolution, which is therefore estimated at 1,728 points. The total number of iterations needed to run the HPT cases is estimated to be 3 million. This is again based on our previous experience of DNS and LES of similar cases that show that we require around 12 flow-through times in order to achieve fully-converged statistics. Spanwise averaging will be employed to help accelerate the convergence.

For the HPT stage case S1 to be conducted on Summit, only one of the rotors will have a rough surface and therefore require a fine O-type grid, noting that the stator and the second rotor (smooth blades) can have coarser O-type grids than that of the roughened rotor. Note that one stator and two rotors are required for each stage configuration (Fig. 3). Therefore the grid for S1 is estimated to be $\times 1.5$ of that needed for cases T1-T4. For the HPT stage cases S2 and S3, 3 finely-resolved O-type grids are required, therefore the estimated in-plane grid count is ≈ 10 million grid points ($\times 3$ that needed for cases T1-T4). In addition a larger number of 10 million iterations is estimated for convergence for the stage cases S1-S3 in order to capture a sufficient number of periods of the wakes' passing frequency.

Table 2. Grid requirements for the proposed runs

Case	**In-plane points** (10^6)	**Spanwise points**	**Total points** (10^9)	**Time step** 10^{-6}	**Iterations** (10^6)
(a) Variation of localized roughness on HPT (Summit)					
T1	3.4	1,728	5.88	3.0	3.0
T2	3.4	1,728	5.88	3.0	3.0
T3	3.4	1,728	5.88	3.0	3.0
T4	3.4	1,728	5.88	3.0	3.0
(b) HPT stage configurations (Summit & Frontier)					
S1	5.0	1,728	8.64	3.0	10
S2	10.2	1,728	17.6	3.0	10
S3	10.2	1,728	17.6	3.0	10

Based on the estimated grid resolution and iteration counts presented in Table 2, the estimated computing resources required for the 7 cases planned over the 12-month period are summarized in Table 3. The required resources is based on performance data from our previous INCITE 2021 grant with a rough surface and a similar Reynolds number. The node count for each simulation will be scaled accordingly to ensure an equal loading and constant point count per node. From our previous scaling testing on Summit, listed in section 3.3, we found 25.2 million points per node (4.2 million per GPU) was the most efficient. However it should be noted that this testing was conducted for previous smooth surface cases. Introduction of the immersed boundary method [45], which will be used in this proposal to incorporate the roughened micro-geometry, means there is a changed computational loading, and from our INCITE 2021 work (where the immersed boundary was used to model a rough surface), we have found that approximately 10 million points per node ($\approx$ 1.7 million points per GPU) is an efficient use of resources, being that which we plan for in table 3. Based on this scaling it will take approximately 0.1 seconds to run one full time step (5 Runge–Kutta cycles) for the HPT. **The time required to conduct the planned simulations is $\mathbf{1,376,000}$ node hours**.

Table 3. Total Requested Computational Time ($\mathbf{1,376,000}$ Node Hrs)

Case	**Total # points** (10^9)	**Timesteps** (10^6)	**Approx. time per step (seconds)**	**Wall-time hours**	**Typical # of nodes**	**Summit node hrs** (10^5)	**Storage (TB)**
Summit							
T1	5.88	3.0	0.1	83	576	0.48	25
T2	5.88	3.0	0.1	83	576	0.48	25
T3	5.88	3.0	0.1	83	576	0.48	25
T4	5.88	3.0	0.1	83	576	0.48	25
S1	8.64	10	0.1	278	806	2.24	30
Sub Total SUMMIT						**4.16**	**130**
Frontier							
S2	17.6	10	0.1	278	1728	4.80	50
S3	17.6	10	0.1	278	1728	4.80	50
Sub Total FRONTIER						**9.60**	**100**
Total						**13.76**	**230**

The request for long term storage is also shown in table 3. The outputs of our cases will be time dependent snapshots of three-dimensional arrays in order to visualize and analyze the flow field. It is estimated that one whole flow field from the HPT production runs will be around 700 GB. By focusing on selective regions, the output of one snapshot can be reduced to around $100-200$ GB depending on the case. A selection of both the full flow field and subsections will be saved for visualization and analysis. For the HPT stage cases (S1-S3) more snapshots are required to capture the motion of the wake and to enable phase lock averaging. In addition to the instantaneous data, statistical three-dimensional arrays will need to be stored. One typical statistics file can be as large as $2,000$ GB. Overall, our total request for long term storage is 230 TB storage space for the 7 proposed cases.

We want to emphasize that the proposed simulations will benefit significantly from the code performance improvement achieved on extensively GPU-accelerated platforms like Summit. The planned production runs will use in excess of 500 nodes. Table 4 suggests the sequence of runs. The proposed sequence is sustainable as it allows balancing the computational load, maintaining a usage of 70,000-80,000 NHrs/month on Summit and 180,000-200,000 NHrs/month (equivalent) on Frontier over the course of the year. By performing these planned runs, we will have an excellent opportunity to contribute to a better understanding of a range of

Table 4. Computational Effort Time Plan (monthly)

Case	**Node Hrs** (10^5)	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**
Summit: Variation of localized roughness on HPT & ×1 HPT stage configuration													
T1	0.48	0.48											
T2	0.48	0.24	0.24										
T3	0.48		0.48										
T4	0.48			0.48									
S1	2.24				1.00	1.00	0.24						
Frontier: ×2 HPT stage configurations													
S2	4.80					1.00	1.00	1.00	1.00	0.80			
S3	4.80						0.80	1.00	1.00	1.00	1.00		

fundamental flow physics problems and impact future gas turbine design. In addition, the fundamental nature of the proposed studies ensures transferable insight to other settings where turbulent flows are confronted with the combination of surface roughness, curvature and strong streamwise pressure gradients.

3.2 Computational Approach

HiPSTAR (High-Performance Solver for Turbulence and Aeroacoustics Research) is a highly accurate structured multi-block compressible fluid dynamics code in curvilinear/cylindrical coordinates, written in FORTRAN. It was developed to conduct cutting-edge DNS, thus can solve the fully nonlinear, time-dependent and three-dimensional Navier-Stokes equations without any empirical assumptions on today's high performance computing (HPC) systems.

To enable world-leading simulations, the numerical algorithm was designed to satisfy several stringent requirements, including resolution of flow features with minimal amplitude and phase errors, stability of the scheme, efficiency of the scheme (i.e. high ratio of accuracy to computational cost) and high parallel efficiency on HPC systems. These requirements are met by using novel wavenumber-optimized compact finite difference schemes (implicit in space, requiring solution of penta-diagonal matrix systems [52]), for some cases in conjunction with a spectral method (using fast Fourier transforms), ultra-low storage frequency optimized explicit Runge-Kutta time integration [53], and by optimizing the code for (low) memory use [54], in light of the severe bandwidth limitations imposed by current multi-core architectures. Furthermore, skew-symmetric splitting is used to improve stability and avoid aliasing errors arising from calculation of the nonlinear convective terms of the compressible Navier–Stokes equations. A formulation is used that splits cubic and quadratic terms differently [55] and is therefore appropriate for compressible flow. Moreover, an efficient turbulence inflow generation technique, based on a digital filter approach, can be used to study the effect of incoming turbulence fluctuations [56]. Of particular importance to the proposed project is the application of an immersed boundary method suited to compressible flow simulations [45] that is able to accurately reproduce the roughness micro-geometry on GT blades. This boundary data immersion method has already been proven to be both accurate and efficient for GT flows with surface roughness in our previous LPT simulations [23] and for HPT more recently [24].

For cases with relatively complex geometries, the discretization benefits from the overset method [51]. As shown in Fig. 3 for the stage configuration (proposed cases S1-S3), the newly introduced overset configuration includes two background H-type grids, which allow for pitchwise periodic boundary conditions, and O-type grids around the blades, enabling adequate resolution around the blade boundaries especially at leading and trailing edges. The H-type and O-type grids overlap with each other, and at the overlapping boundaries, continuity conditions are imposed as variables are interpolated with a fourth-order Lagrangian method and communicated between the grids. The principal advantage of this overset configuration is that the mesh generation process for the complex geometry of an HPT is straightforward, as the orthogonality of the computational grid is easy to enforce. Furthermore, a novel sliding grid method [57], which was demonstrated to be highly accurate for turbomachinery stator-rotor interaction and similar problems, will be applied in the proposed cases to simulate the wake-blade interactions.

Two different parallelization strategies are available for different types of parallel HPC systems. For CPU platforms, a hybrid OMP/MPI parallelization can be applied. The hybrid OMP/MPI parallelization was first implemented through a Cray Center of Excellence project (Dr. Tom Edwards). In this project, HiPSTAR was adapted to use hybrid parallelism using the OpenMP API. The domain decomposition is such that the first two (in-plane) directions are split up into MPI subdomains and the third direction is parallelized using OpenMP. This allows for efficient parallelization also when using the spectral Fourier method in the third direction. Furthermore, a novel approach in efficiently parallelizing compact difference schemes was implemented [52], avoiding traditionally used global data transposes or MPI all-to-all calls.

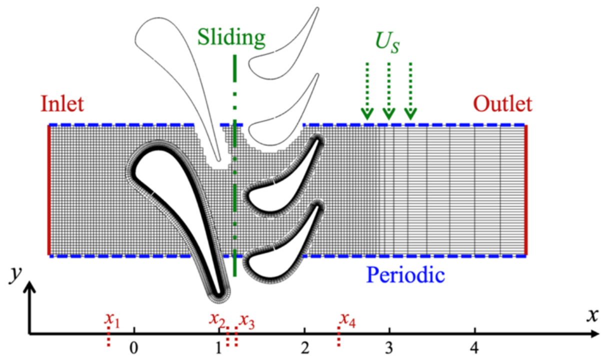

Figure 3. Overset grid configuration previously used for GT stage configurations, e.g. smooth HPT in [30].

For GPU accelerated systems, HiPSTAR has been ported using OpenACC. In 2013, HiPSTAR was subject to an ORNL Director's Grant (ARD015), in which John Levesque (Cray Research) and Ramanan Sankaran (ORNL) ported the MPI/OMP code to OpenACC. Without significantly altering the structure of the code, the GPU version slightly outperformed the hybrid OMP/MPI version. In a follow-on ORNL Director's Grant for 2014 (ARD106), further modifications to HiPSTAR-G implemented by Dr. Richard Pichler with support by NVIDIA and CRAY resulted in a speed-up of over 50% over the CPU version. Furthermore, the code was restructured during 2016/2017 by Dr. Richard Pichler to generate fewer larger compute kernels which has given another significant boost to performance, such that the OpenACC version reduced the time per time step by a factor of four with respect to the hybrid MPI/OpenMP version on Titan (i.e. comparing one K20x with one 16 core AMD Opteron). During the GPU-Hackathons held by ORNL and Pawsey in Australia in April 2018 and March 2019, part of the team on this proposal re-optimized most of the computationally expensive kernels using asynchronization strategies, and the performance of HiPSTAR on the NVIDIA P100 GPUs was further improved.

3.3 Parallel Performance

In preparation for our Early Science allocation that commenced in January 2019, HiPSTAR was ported and further optimized for the IBM/NVIDIA V100 architecture on Summit. The NVIDIA V100 GPU accelerator on Summit shows to be very powerful running HiPSTAR, and we get a factor of 1.5 $\sim$ 2 acceleration with a single-GPU case compared to the previous generation NVIDIA GPU P100. Furthermore, a case with around 0.1 billion grid points was tested separately on 6 GPU accelerators (1 Summit node) and 384 CPUs (approximately 10 Summit nodes), showing that HiPSTAR demonstrates a performance improvement of a factor of 2.5 $\sim$ 3 running on GPUs, compared to using the CPUs on ten times the number of nodes.

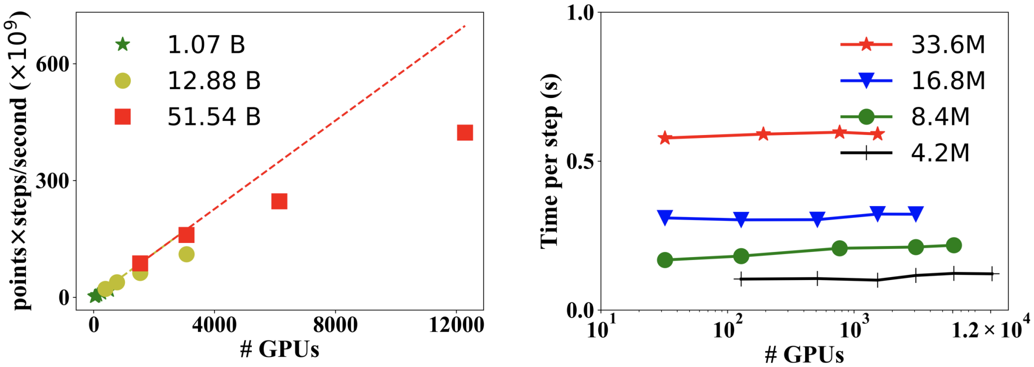

Figure 4. Left: Strong scaling on Summit for three test cases, with 1.07 (green), 12.88 (yellow) and 51.54 billion grid points (black). Symbols represent measurements, dashed lines linear scaling based on the lowest tested node count at that grid size. Right: Weak scaling, increasing GPU count with constant grid points on each GPU, with 33.6 (red), 16.8 (blue), 8.4 (green) and 4.2 (black) million grid points on each GPU. The largest job has around 51 billion grid points running on 12, 288 GPUs (2, 048 Summit nodes).

Furthermore, parallel scaling of HiPSTAR on Summit was extensively tested, and the results are presented in Fig. 4. A series of tests, with the total number of grid points varying from 0.27 to 51.5 billion, were performed to show the strong and weak scaling of the code. For strong scaling in the left figure, three cases with different numbers of total points are presented, using grid points per unit wall clock time as a performance metric and the GPU count as the independent variable. The line showing ideal (linear) scaling is defined separately for each grid size (extrapolating from the result obtained at the lowest GPU count in each case). Data points related to the performance of the larger simulations lie on or near the extrapolated ideal scaling line of the smaller cases. It can be seen from the figure that the code shows reasonably good scaling for all tested cases. In particular, the parallel efficiency remains above 60% when the number of points per GPU is more than 4.2 million (25.2 million per node, which corresponds to our planned production runs. As noted above, this testing was conducted for smooth-wall cases, and the immersed boundary method required to incorporate the roughened geometry, means there is a changed computational load balance, for which we have found $\approx$ 1.7 million points per GPU to offer good scaling.

Finally, weak scaling is shown in the right figure, with the number of points on each GPU kept constant. We can see wall clock time per time step stays almost constant with larger cases, up to over 12, 000 GPUs (2, 000 Summit nodes). Considering the good weak scaling characteristics of the code, it is therefore straightforward to scale to large domain sizes provided that the work performed per GPU is kept constant. Thus, everything is in place to conduct highly computationally efficient high-fidelity simulations of the proposed cases in GPU mode on Summit, and this benchmark test can be used to estimate the wall clock times for the current proposed project.

3.4 Developmental Work

The algorithms, flow configurations, and parallelization strategies of HiPSTAR have been extensively tested and optimized for turbomachinery flows, and were already used in our previous INCITE (TUR105), ALCC (TUR123) projects. In particular the ability to use the immersed boundary method to realize small-scale textured surfaces at realistic flow conditions on a GT blade (Fig. 2), thus being used at scale, has been demonstrated with the INCITE2021 project and a current INCITE (ENG112) allocation looking at riblets on compressor blades.

For cases T1-T4 & S1 of the proposed project to be run on Summit, no extra development efforts will be needed, except for possible minor performance enhancements as the Summit software stack gets updated over the duration of the INCITE project. For cases S2-S3, GPU-offloading in HiPSTAR is actively being translated to OpenMP GPU-offloading which will be better supported on the proposed programming environment on Frontier. Additionally, this effort will ensure the future portability and future-proofing of HiPSTAR. Moreover, the most computationally-heavy back-end routines are currently additionally coded in CUDA for improved performance on NVIDIA GPUs on which HiPSTAR has been run to date. In preparation for Frontier and the associated production-level GPU compute capacity, these back-end routines will be reproduced in the HIP API for optimised performance on the AMD GPUs of Frontier. Our efforts on this front are supported by our PaCER (Pawsey Centre for Exascale Readiness) project with the Pawsey Supercomputing Centre in Perth, Australia. Testing of this translation from OpenACC/CUDA (as used to date) to OpenMP/HIP (for use on the Frontier AMD system) is currently being undertaken on an AMD GPU test node at the Pawsey Centre. Members of the assembled team from the University of Melbourne will participate in a mentoring program facilitated by Pawsey in July 2022, where we will work with AMD specialists to complete the necessary porting and performance testing. This will ensure HiPSTAR is ready for Frontier and the allocation is most effectively used.

References

[1] J. P. Bons. A review of surface roughness effects in gas turbines. *Journal of Turbomachinery*, 132(2), 2010.

[2] M. Lorenz, A. Schulz, and H.-J. Bauer. Experimental study of surface roughness effects on a turbine airfoil in a linear cascade—part I: External heat transfer. *Journal of turbomachinery*, 134(4), 2012.

[3] V. Michelassi, F. Martelli, R. Dénos, T. Arts, and C. H. Sieverding. Unsteady heat transfer in stator-rotor interaction by two equation turbulence model. *Journal of Turbomachinery*, 121(3), 1999.

[4] R. Pichler, R. D. Sandberg, V. Michelassi, and R. Bhaskaran. Investigation of the accuracy of RANS models to predict the flow through a low-pressure turbine. *Journal of Turbomachinery*, 138(12):121009, 2016.

[5] J. Joo, G. Medic, and O. Sharma. Large-eddy simulation investigation of impact of roughness on flow in a low-pressure turbine. In *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*. ASME, 2016.

[6] R. D Sandberg and V. Michelassi. Fluid dynamics of axial turbomachinery: Blade-and stage-level simulations and models. *Ann. Rev. Fluid Mech.*, 54:255–285, 2022.

[7] J.-C. Han, S. Dutta, and S. Ekkad. *Gas turbine heat transfer and cooling technology*. CRC press, 2012.

[8] V. Michelassi, L. W. Chen, R. Pichler, and R. D. Sandberg. Compressible Direct Numerical Simulation of low-pressure turbines. Part II: effect of inflow disturbances. *Journal of Turbomachinery*, 137(7):071005, 2015.

[9] R. D. Sandberg, V. Michelassi, R. Pichler, L. W. Chen, and R. Johnstone. Compressible direct numerical simulation of low-pressure turbine. Part I: methodology. *Journal of Turbomachinery*, 137(5):051011, 2015.

[10] A. Wheeler, R. D. Sandberg, N. D. Sandham, R. Pichler, V. Michelassi, and G. Laskowski. Direct numerical simulations of a high-pressure turbine vane. *Journal of Turbomachinery*, 138(7):071003, 2016.

[11] R. D. Sandberg and V. Michelassi. The current state of high-fidelity simulations for main gas path turbomachinery components and their industrial impact. *Flow, Turbulence and Combustion*, pages 1–52, 2019.

[12] T. O. Jelly and A. Busse. Reynolds and dispersive shear stress contributions above highly skewed roughness. *Journal of Fluid Mechanics*, 852:710–724, 2018.

[13] J. Jiménez. Turbulent flows over rough walls. *Annual Review of Fluid Mechanics*, 36:173–196, 2004.

[14] M. Stripf, A. Schulz, and H.-J. Bauer. Modeling of rough-wall boundary layer transition and heat transfer on turbine airfoils. *Journal of Turbomachinery*, 130(2), 2008.

[15] T. Ikeda and P. A. Durbin. Direct simulations of a rough-wall channel flow. *Journal of Fluid Mechanics*, 571:235–263, 2007.

[16] T. O. Jelly, R. C. Chin, S. J. Illingworth, J. P. Monty, I. Marusic, and A. Ooi. A direct comparison of pulsatile and non-pulsatile rough-wall turbulent pipe flow. *Journal of Fluid Mechanics*, 895:R3, 2020.

[17] J. H. Lee, H. J. Sung, and P.-Å. Krogstad. Direct numerical simulation of the turbulent boundary layer over a cube-roughened wall. *Journal of Fluid Mechanics*, 669:397–431, 2011.

[18] S. Leonardi and I. P. Castro. Channel flow over large cube roughness: a direct numerical simulation study. *Journal of Fluid Mechanics*, 651:519–539, 2010.

[19] L. Chan, M. MacDonald, D. Chung, N. Hutchins, and A. Ooi. A systematic investigation of roughness height and wavelength in turbulent pipe flow in the transitionally rough regime. *Journal of Fluid Mechanics*, 771:743–777, 2015.

[20] I. Marusic, R. Mathis, and N. Hutchins. Predictive model for wall-bounded turbulent flow. *Science*, 329(5988):193–196, 2010.

[21] J. E. Dees and D. G. Bogard. Effects of regular and random roughness on the heat transfer and skin friction coefficient on the suction side of a gas turbine vane. *Journal of Turbomachinery*, 130(4), 2008.

[22] M. N. Goodhand, K. Walton, L. Blunt, H. W. Lung, R. J. Miller, and R. Marsden. The limitations of using Ra to describe surface roughness. *Journal of Turbomachinery*, 138(10), 2016.

[23] F. Hammer, N. D. Sandham, and R. D. Sandberg. Large eddy simulations of a low-pressure turbine – roughness modeling and the effects on boundary layer transition and losses. In *ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition*, pages GT2018–75796. ASME, 2018.

[24] T.O. Jelly, M. Nardini, M. Rosenzweig, J. Leggett, I. Marusic, and R.D. Sandberg. High-fidelity computational study of roughness effects on high pressure turbine performance and heat transfer. In *12th International Symposium on Turbulence and Shear Flow Phenomena (TSFP12), Osaka, Japan, July 19-22*, 2022.

[25] R. Pichler, R. D. Sandberg, and V. Michelassi. Assessment of grid resolution requirements for accurate simulation of disparate scales of turbulent flow in low-pressure turbines. In *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*. ASME, 2016.

[26] R. Pichler, R. D. Sandberg, G. Laskowski, and V. Michelassi. High-fidelity simulations of a linear HPT vane cascade subject to varying inlet turbulence. In *ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition*. ASME, 2017.

[27] J. Leggett, S. Priebe, A. Shabbir, V. Michelassi, R. D. Sandberg, and E. S. Richardson. LES loss prediction in an axial compressor cascade at off-design incidences with free stream disturbances. *Journal of Turbomachinery*, 7(140):071005, 2018.

[28] R. Pichler, J. Kopriva, G. Laskowski, V. Michelassi, and R. D. Sandberg. Highly resolved LES of a linear HPT vane cascade using structured and unstructured codes. In *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*. ASME, 2016.

[29] Y. Zhao and R. D. Sandberg. High-fidelity simulations of a high-pressure turbine vane subject to large disturbances: effect of exit Mach number on losses. *ASME J. Turbomachinery*, 143(9), 2021.

[30] Y. Zhao and R. D. Sandberg. High-fidelity simulations of a high-pressure turbine stage: effects of Reynolds number and inlet turbulence. In *Turbo Expo: Power for Land, Sea, and Air*, volume 84911, page V02BT32A004. American Society of Mechanical Engineers, 2021.

[31] P. A. Durbin and X. Wu. Transition beneath vortical disturbances. *Annual Review of Fluid Mechanics*, 39:107–128, 2007.

[32] R. G. Jacobs and P. A. Durbin. Simulations of bypass transition. *Journal of Fluid Mechanics*, 428:185–212, 2001.

[33] L. Brandt, P. Schlatter, and D. S. Henningson. Transition in boundary layers subject to free-stream turbulence. *Journal of Fluid Mechanics*, 517:167–198, 2004.

[34] H. H. Fernholz and D. Warnack. The effects of a favourable pressure gradient and of the Reynolds number on an incompressible axisymmetric turbulent boundary layer. Part 1. The turbulent boundary layer. *Journal of Fluid Mechanics*, 359:329–356, 1998.

[35] D. Warnack and H. H. Fernholz. The effects of a favourable pressure gradient and of the Reynolds number on an incompressible axisymmetric turbulent boundary layer. Part 2. The boundary layer with relaminarization. *Journal of Fluid Mechanics*, 359:357–381, 1998.

[36] S. Nagarajan, S. K. Lele, and J. H. Ferziger. Leading-edge effects in bypass transition. *Journal of Fluid Mechanics*, 572:471–504, 2007.

[37] N. Hutchins and I. Marusic. Evidence of very long meandering features in the logarithmic region of turbulent boundary layers. *Journal of Fluid Mechanics*, 579:1–28, 2007.

[38] P. A. Durbin, G. Medic, J.-M. Seo, J. K. Eaton, and S. Song. Rough wall modification of two-layer k-ε. *Journal of Fluids Engineering*, 123(1):16–21, 2001.

[39] T. Knopp, B. Eisfeld, and J. B. Calvo. A new extension for k–ω turbulence models to account for wall roughness. *International Journal of Heat and Fluid Flow*, 30(1):54–65, 2009.

[40] K. Duraisamy, G. Iaccarino, and H. Xiao. Turbulence modeling in the age of data. *Annual Review of Fluid Mechanics*, 51:357–377, 2019.

[41] M. A. Jouybari, J. Yuan, and G. J. Brereton. Data-driven high-fidelity prediction of the equivalent sand-grain height of rough surfaces. *arXiv preprint arXiv:2002.01515*, 2020.

[42] Y. Zhao and R. D. Sandberg. Using a new entropy loss analysis to assess the accuracy of RANS predictions of an HPT vane. In *ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition*. ASME, 2019.

[43] J. Leggett, E. Richardson, S. Priebe, A. Shabbir, V. Michelassi, and R. Sandberg. Loss analysis of unsteady turbomachinery flows based on the mechanical work potential. *Journal of Turbomachinery*, 142(11), 2020.

[44] R. J. Miller. Mechanical work potential. In *ASME Turbo Expo 2013: Turbine Technical Conference and Exposition*. ASME, 2013.

[45] S. C. Schlanderer, G. D. Weymouth, and R. D. Sandberg. The boundary data immersion method for compressible flows with application to aeroacoustics. *Journal of Computational Physics*, 333:440–461, 2017.

[46] D. Lengani, D. Simoni, R. Pichler, R. D. Sandberg, V. Michelassi, and F. Bertini. On the identification and decomposition of the unsteady losses in a turbine cascade. *Journal of Turbomachinery*, 141(3), 2019.

[47] J. Leggett, Y. Zhao, E. Richardson, and R.D. Sandberg. Turbomachinery loss analysis: the relationship between mechanical work potential and entropy analyses. In *ASME IGTI*, pages GT2021–59436. ASME, 2021.

[48] R. Dutta, J. Nicolle, A. M. Giroux, and U. Piomelli. Evaluation of turbulence models on roughened turbine blades. In *IOP Conference Series: Earth and Environmental Science*, volume 49, page 062007. IOP Publishing, 2016.

[49] J. Weatheritt and R. D. Sandberg. A novel evolutionary algorithm applied to algebraic modifications of the RANS stress–strain relationship. *Journal of Computational Physics*, 325:22–37, 2016.

[50] Y. Zhao, H. Akolekar, J. Weatheritt, V. Michelassi, and R. D. Sandberg. RANS turbulence model development using CFD-driven machine learning. *Journal of Computational Physics*, 411(109413), 2020.

[51] M. Deuse and R. D. Sandberg. Implementation of a stable high-order overset grid method for high-fidelity simulations. *Computers & Fluids*, 211:104449, 2020.

[52] J. W. Kim and R. D. Sandberg. Efficient parallel computing with a compact finite difference scheme. *Computers & Fluids*, 58:70–87, 2012.

[53] C. A. Kennedy, M. H. Carpenter, and R. M. Lewis. Low-storage, explicit Runge–Kutta schemes for the compressible Navier–Stokes equations. *Applied Numerical Mathematics*, 35(3):177–219, 2000.

[54] R. D. Sandberg. Direct numerical simulations for flow and noise studies. *Procedia Engineering*, 61:356–362, 2013.

[55] C. A. Kennedy and A. Gruber. Reduced aliasing formulations of the convective terms within the Navier–Stokes equations for a compressible fluid. *Journal of Computational Physics*, 227(3):1676–1700, 2008.

[56] E. Touber and N. D. Sandham. Large-eddy simulation of low-frequency unsteadiness in a turbulent shock-induced separation bubble. *Theoretical and Computational Fluid Dynamics*, 23(2):79–107, 2009.

[57] R. Johnstone, L. W. Chen, and R. D. Sandberg. A sliding characteristic interface condition for direct numerical simulations. *Computers & Fluids*, 107:165–177, 2015.

PERSONNEL JUSTIFICATION AND MANAGEMENT PLAN

PERSONNEL JUSTIFICATION

The proposed effort will involve nine main scientists from the University of Melbourne and General Electric (GE) Aviation: at the University of Melbourne, Prof. Richard Sandberg will lead the research, Prof. Ivan Marusic will provide insights on the turbulent boundary layer modelling and its interaction with rough surfaces, and Drs Melissa Kozul, Pawel Przytarski, John Leggett and Massimiliano Nardini will perform the bulk of the planned simulations. Drs Sriram Shankaran and Aamir Shabbir will be the main GE personnel supporting the project, while additional support from two Chief Consulting Engineers (Dr. William Solomon and Mr. Paul Vitt) from GE Aviation will assure the relevance of the configurations chosen, and help results feed straight into GE's design process.

Prof. R. D. Sandberg (RDS) is Chair of Computational Mechanics in the Department of Mechanical Engineering at the University of Melbourne. He has extensive experience in conducting large-scale flow simulations, in particular direct numerical simulations (DNS). RDS received his PhD at the University of Arizona with Prof Hermann Fasel in 2004 where he used DNS to study hydrodynamic instabilities in supersonic base flows. He held a RAEng/EPSRC Fellowship from 2007-2012, during which he developed the compressible high-performance DNS solver HiPSTAR, optimized for high-performance computing. In a large UK research council funded collaborative project between Cambridge and Southampton on the universality of fine-scale turbulence, RDS coordinated all computational work. RDS lead the UK Turbulence Consortium (UKTC), between 2011 and 2015, coordinating the work packages for compressible flows and flow visualizations and databases, and he sat on the EPSRC computing resource allocation panel and was on the Management Committee of CCP12: High Performance Computing in Engineering. In 2015, he moved to the University of Melbourne in Australia, taking on the role as Chair of Computational Mechanics. He has recently been awarded an Australian Research Council Future Fellowship (2020-2024). Between 2016 and 2019 he was member of the Australian National Computational Merit Allocation Committee. Over the last three years he has been awarded several hundred million core hours (using GPU acceleration) for high-fidelity turbomachinery simulations through INCITE, ALCC and the Swiss CSCS computing centre. He is an editor for *Flow, Turbulence and Combustion*, *Theoretical and Computational Fluid Dynamics* and a member of ASME's Gas Turbine Technology Group (GTTG) leadership team.

Prof. I. Marusic (IM) is a Redmond Barry Distinguished Professor in the Faculty of Engineering & IT at the University of Melbourne. IM is recognised internationally as a leader in the area of experimental fluid mechanics and turbulent flows. He received his PhD in 1992 and BE (Hons) Mech in 1987 from the University of Melbourne. From 1998-2006 he was a faculty member at the University of Minnesota in the Department of Aerospace Engineering and Mechanics, and returned to Australia at the end of 2006. IM's research is primarily in experimental and theoretical studies of turbulence at high Reynolds numbers. This includes studies in atmospheric surface layer flows and aquatic ecosystems. Over his career he has held a number of prestigious fellowships, including an Australian Research Council (ARC) Laureate Fellowship (2012-2017), ARC Federation Fellowship (2006-2011), and a Packard Fellowship in Science and Engineering (2001-2006). He has been an Associate Editor for the *Journal of Fluid Mechanics* since 2008, and was the augural President and Founding member of the Australasian Fluid Mechanics Society. IM is recipient of a number of awards including the Woodward Medal from the University of Melbourne and the Stanley Corrsin Award from the American Physical Society. He is a Fellow of the American Physical Society, the Australasian Fluid Mechanics Society and the Australian Academy of Technology and Engineering. In 2014 IM was elected as a Fellow of the Australian Academy of Science and is currently serving as a Council Member of the Academy.

Dr. Sriram Shankaran (SS) is a Mechanical Engineer at the Aero-Thermal Technologies Division at General Electric Aviation, based in Evendale, OH. He is responsible for the technical development of methods that leverage CFD-based tools in the aerodynamic and thermal design of the new Aviation engine products. He has a background in computational methods and obtained his PhD from Stanford University working on developing numerical methods for the analysis and design of aircraft and sailing geometries. Dr. Shankaran served as Co-PI for the ENG111 project under a 2020 INCITE grant.

Dr. Aamir Shabbir (AS) is a Principal Engineer at GE Aviation in Cincinnati. He has worked in the area of CFD of turbomachinery and turbulence modeling for 28 years during which time his responsibilities have included: developing turbulence models for turbomachinery applications, developing CFD codes and tools, establishing CFD best practices, and using CFD to understand turbomachinery flow physics. Aamir has supported the design of fans, boosters, and compressors during his tenure at the engine companies. He was part of the team that won the NASA Group Achievement Award for the design of boundary layer ingesting fan stage. He was co-winner of the Best Paper of the Year award from the American Society of Mechanical Engineers in 2005. He is co-developer of one of the turbulence models that is available in the commercial CFD code FLUENT. He has also closely worked with Prof. Sandberg's group over last three years in the use of LES for understanding compressor losses. Aamir obtained his Ph.D. from the State University of New York at Buffalo in 1987.

Dr Melissa Kozul (MK), is a postdoctoral research fellow in the Department of Mechanical Engineering at the University of Melbourne. She completed her PhD at the University of Melbourne in 2018 and her research expertise is in the high-fidelity simulations of fundamental turbulent flows that feature critically in energy and transport technologies. To date her research interests have included wall-bounded flows, homogeneous isotropic turbulence and turbulent multiphase flows. She has extensive experience in designing and executing massively parallel simulations and is currently collaborating with the Pawsey Supercomputing Centre in Perth, Australia, to ready the compressible high-performance DNS solver HiPSTAR (as used in this proposal) for next-generation supercomputing architectures as part of the Pawsey Centre for Extreme Scale Readiness (PaCER) scheme. She currently leads the 2022 INCITE (ENG112) simulation campaign on turbomachinery flows.

Dr Pawel Przytarski (PP), is a postdoctoral research fellow in the Department of Mechanical Engineering at the University of Melbourne. He completed his PhD at the Whittle Laboratory, University of Cambridge, where he studied loss mechanisms in compressor flows using high-fidelity simulations. During his PhD he co-wrote multiple UK Turbulence Consortium and EPSRC project proposals and set up a collaboration with the University of Genoa. His work into multi-stage compressor turbulence production mechanisms was recognized by the UK Turbulence Consortium earning him Timothy Bruce Nickels award and by Rolls-Royce who invited him to present at their Annual Review. He continues his research into compressor flows and is currently responsible for running large-scale compressor stage simulations as part of the PRACE project. In addition, he has recently secured Maria Sklodowska-Curie Global Fellowship funded by the European Commission to further explore how high-fidelity data can be used to better inform industrial flow modelling by employing data-driven techniques. He has participated in the 2021 and current 2022 INCITE (ENG112) projects on turbomachinery flows.

Dr John Leggett (JL), is a postdoctoral research fellow in the Department of Mechanical Engineering at the University of Melbourne. His research interests are in high fidelity numerical simulation of gas turbine machinery, focusing on Large Eddy Simulations and loss model development. His work has focused largely on high pressure compressors, investigating the effects of incidence, free-stream turbulence and full stage wake blade interactions, but recently also on loss analysis if HPT stage simulations data. He has worked on code development and optimization to improve performance for both CPU and hybrid architecture as well as implementation of new functionality. He has long-term experience in High Performance Computing, Large Eddy Simulations, Data Management, loss analysis and axial compressor and turbine aerodynamics. He also participated in the 2020 and 2021 INCITE projects on turbomachinery flows.

Dr. Massimiliano Nardini (MN), is a postdoctoral research fellow in the Department of Mechanical Engineering at the University of Melbourne. His main research interest is in computational fluid dynamics, with a focus on high-fidelity simulations of compressible turbulent flows applied to aeroacoustics, turbomachinery and fluid-structure interaction. He has particular expertise in the immersed boundary method, and developed an efficient, unsteady three-dimensional Boundary Data Immersion Method (BDIM) within the main research code HiPSTAR to simulate complex moving and non-moving geometries immersed in a flow. His immersed boundary code developments have been successfully employed to simulate small-scale surface roughness on high-pressure turbine blades as part of our 2021 INCITE, and engineered micro-grooves (riblets) on high-pressure compressor blades as part of our current 2022 INCITE (ENG112) project. He actively participates in the current 2022 INCITE project on turbomachinery flows, working on the further development of the immersed boundary method, as well as developing frameworks for the output/processing of on-surface data as relevant to turbomachinery flows.

The team is rounded out by the following collaborators from GE Aviation:

Dr William Solomon (WS), is Chief Consulting Engineer for Compression Aerodynamics at GE Aviation in Cincinnati, responsible for Fan and Compressor Aerodynamic Design. WS is a discipline leader with responsibility for the majority of GE Commercial Programs and his roles include technical mentoring, training and leading design reviews. WS completed his PhD in 1997 on unsteady boundary layer transition in compressors and spent 3 years at the Ohio Aerospace institute further investigating boundary layer transition, before joining GE in 2000. Over the last 20 years, he has been involved in and lead numerous compressor projects. Of particular relevance to this proposal is his expertise on surfaced roughness effects in the context of compressors. In fact, he has completed a GE review of the impact of airfoil surface roughness on key compressor product performance data with comparison to proprietary modeling.

Paul Vitt (PV), is a Chief Consulting Engineer for Turbine Aerodynamics at GE Aviation, based in Evendale, OH. He is responsible for design of turbine airfoils across the full range of products, including high- and low pressure- turbines for turbofan, turbojet and turboshaft engines, and aero-derivative applications. He has worked in the gas turbine field for 23 years, holds 18 patents for gas turbine engines, and has authored 6 peer-reviewed technical papers.

Expertise

The table below summarizes the expertise of the entire team by discipline. The team has proven expertise in turbomachinery design and performance prediction, fundamental gas-dynamics and turbulence, roughness, lower order models, machine-learning (ML), high-fidelity simulations, and massive parallel computations from several past and existing research and computing time grants in both the USA, the EU and Australia. The team blends the fundamental research capabilities from the University of Melbourne with the industrial and applied research perspective from General Electric.

	RDS	IM	SS	AS	MK	PP	JL	MN	WS	PV
T/M aero-thermodynamics			■	■		■			■	■
Turbines	■		■	■		■	■		■	
Rough surfaces		■						■	■	■
Turbulent flows	■	■	■	■	■	■	■	■	■	■
Data-driven modeling/ML	■					■				
High-fidelity Simulations	■		■	■	■	■	■	■	■	
Parallel Computation (CPU)	■		■		■	■	■	■	■	
Parallel Computation (GPU)	■				■		■	■		

MANAGEMENT PLAN

The decision-making process is based upon a successful and long-lasting cooperation between the University of Melbourne and General Electric. The fruitful collaboration between the University team and General Electric has produced numerous invited lectures, journal papers, and peer-reviewed conference papers. The technical plan described in the proposal is the outcome of a detailed discussion between Industry and Academia, and we do not expect substantial deviations. RDS, the principal developer of HiPSTAR, will overlook the simulations to ensure accuracy and rigor and IM will provide guidelines and assist in postprocessing data related to turbulent boundary layers and their interaction with textured surfaces. The set-up and run execution will be completed by MK, PP, JL and MN, who have all conducted simulations on Summit as part of previous INCITE projects. The investigators and collaborators from GE (AS, SS, WS and PV) will ensure the computer simulations address specific design needs and will work with the team to extract turbomachinery relevant data, as detailed in the project narrative.

Proposal Title: Toward In-Service Realism: DNS of Gas Turbine Blades with Localized Roughness

Year 1: (a) Variation of localized roughness on HPT

Milestone:	**Details (as appropriate):**	**Dates:**	**Status: (renewals only)**
M1-1: Case T1	**Resource:** SUMMIT **Node-hours:** 48,000 **Filesystem storage (TB and dates):** 25TB for 4 months **Archival storage (TB and dates):** 10TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** T1 HPT with minimal extent of LE roughness **Dependencies:** None	January	N.A.
M1-2: Case T2	**Resource:** SUMMIT **Node-hours:** 48,000 **Filesystem storage (TB and dates):** 25TB for 3 months **Archival storage (TB and dates):** 10TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** T2 HPT with intermediate extent of LE roughness 1 1**Dependencies:** None	January- February	N.A.
M1-3: Case T3	**Resource:** SUMMIT **Node-hours:** 48,000 **Filesystem storage (TB and dates):** 25TB for 3 months **Archival storage (TB and dates):** 10TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** T3 HPT with intermediate extent of LE roughness 2 **Dependencies:** None	February	N.A.
M1-4: Case T4	**Resource:** SUMMIT **Node-hours:** 48,000 **Filesystem storage (TB and dates):** 25TB for 4 months **Archival storage (TB and dates):** 10TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** S4 HPT with maximal extent of LE roughness **Dependencies:** None	March	N.A.

Year 1: (b) HPT stage configurations			
M2-1: Case S1	**Resource:** SUMMIT **Node-hours:** 224,000 **Filesystem storage (TB and dates):** 30TB for 4 months **Archival storage (TB and dates):** 15TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** S1 stage configuration: rotor 1 rough **Dependencies:** Following the investigation of M1 (cases T1-T4), for the extent of LE roughness considered being indicative of in-service HPT, roughness distributions of interest will be selected for the stage configurations, that is say, cases with significant changes in the location of transition to turbulent flow over the HPT due to the localized roughness distribution. S1 will only have 1 rotor with a roughened surface, with the stator and the 2nd rotor smooth (each stage configuration features one stator and two rotors, see Fig. 3 in Project Narrative). This will not only constrain the cost of this case to be run on Summit, but will allow direct comparison of a smooth and rough rotor seeing the same incoming flow from the stator, allowing isolation of the roughness effects during post-processing.	April-June	N.A.
M2-2: Case S2	**Resource:** FRONTIER **Node-hours:** 480,000 **Filesystem storage (TB and dates):** 50TB for 4 months **Archival storage (TB and dates):** 20TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** S2 stage configuration: stator & both rotors rough, distribution 1 **Dependencies:** Again the localized roughness distributions for the stage configurations S2 and S3 will be informed by the investigation of M1 (cases T1-T4), however in S2 and S3, all three of the stator and two rotors will be roughened. The two rotors could additionally have different roughness distributions, allowing us to study (between S2 and S3) a total of four localized roughness distributions in the stage configuration.	May-September	N.A.
M2-3: Case S3	**Resource:** FRONTIER **Node-hours:** 480,000 **Filesystem storage (TB and dates):** 50TB for 4 months **Archival storage (TB and dates):** 20TB for 3 years **Software Application:** HIPSTAR DNS **Tasks:** S2 stage configuration: stator & both rotors rough, distribution 2 **Dependencies:** As for M2-2	June-October	N.A.

PUBLICATIONS RESULTING FROM INCITE AWARDS

All publications listed below have (partly) resulted from the INCITE AWARDS TUR105 and ENG112.

Journal Publications:

Sandberg, R.D., Michelassi, V. 2022, “Fluid Dynamics of Axial Turbomachinery: Blade- and Stage-Level Simulations and Models”, *Annual Review of Fluid Mechanics*, 54:255-85.

Zhao, Y., Sandberg R.D., 2021 “High-fidelity Simulations of a High-pressure Turbine Vane: Effect of Exit Mach Number on Losses”, *J. Turbomachinery,* 143(9): 091002.

Zhao, Y., Sandberg, R.D., 2020 “Bypass transition in boundary layers subject to strong pressure gradient and curvature effects”, *J. Fluid Mech.*, 888, A4.

Zhao, Y., Akolekar, H. D., Weatheritt, J., Michelassi, V. and Sandberg, R.D., 2020 “Turbulence model development using CFD-driven machine learning”, *J. Comput. Phys.*, 411, 109413.

Zhao, Y., Sandberg, R.D., 2020 “Using a New Entropy Loss Analysis to Assess the Accuracy of RANS Predictions of an HPT Vane”, *J. Turbomachinery*, TURBO-20-1017.

Sandberg, R.D., Michelassi, V., 2019 “The Current State of High-Fidelity Simulations for Main Gas Path Turbomachinery Components and Their Industrial Impact”, *Flow, Turbulence and Combustion*, 102(4)

Michelassi, V., Chen, L., Pichler, R., Sandberg, R.D., Bhaskaran, R. 2016 “High-Fidelity Simulations of Low-Pressure Turbines: Effect of Flow Coefficient and Reduced Frequency on Losses”, *J. Turbomachinery*, 138(11).

Conference Proceedings:

Leggett, J., Zhao, Y., Sandberg, R.D., 2022, “High-fidelity simulation study of the unsteady flow effects on high-pressure turbine blade performance”, ASME IGTI, GT2022-82370

Jelly, T.O., Nardini, M., Rosenzweig, M., Leggett, J., Marusic, I., Sandberg, R.D., 2022, “High-fidelity computational study of roughness effects on high pressure turbine performance and heat transfer”, 12th International Symposium on Turbulence and Shear Flow Phenomena (TSFP12), Osaka, Japan, July 19-22

Zhao, Y., Sandberg, R.D., 2021 “High-fidelity simulations of a high-pressure turbine stage: effects of Reynolds number and inlet turbulence”, ASME IGTI, GT2021-58995.

Zhao, Y., Sandberg R.D., 2021 “High-fidelity simulations of a high-pressure turbine vane with end walls: impact of secondary structures and spanwise temperature profiles on losses”, ASME IGTI, GT2021-58816.

Leggett, J., Zhao, Y., Richardson, E., Sandberg, R.D., 2021 “Turbomachinery loss analysis: the relationship between mechanical work potential and entropy analyses”, ASME IGTI, GT2021-59436.

Leggett, J., Sandberg, R.D., 2021 “Highly resolved simulations of a CDA compressor cascade: effect of Reynolds number on losses”, ASME IGTI, GT2021-58665.

Zhao, Y., Pichler, R., Sandberg, R.D., 2018 “Impact of Large-Scale Structures on the Flow Around a Transonic High-Pressure Turbine Vane”, *APS-DFD, Gallery of Fluid Motion*, V00040.

Pichler, R., Sandberg, R.D., Laskowski, G., Michelassi, V., 2017 “High-Fidelity Simulations of a Linear HPT Vane Cascade Subject to Varying Inlet Turbulence”, *ASME Turbo Expo*, GT2017-63079.

Weatheritt, J., Pichler, R., Sandberg, R. D., Laskowski, G., Michelassi, V. 2017 “Machine learning for turbulence model development using a high-fidelity HPT cascade simulation”, *ASME Turbo Expo*, GT2017-63497.

Pichler, R., Michelassi, V., Sandberg, R.D., Laskowski, G. 2017, “Transition investigations based on large eddy simulation of high-pressure turbines vane at realistic Reynolds and Mach numbers” proceedings of the *ERCOFTAC Workshop Direct and Large-Eddy Simulations 11*.

Pichler, R., Kopriva, J., Laskowski, G., Michelassi, V., Sandberg, R.D., 2016 “Highly Resolved LES of a Linear HPT Vane Cascade Using Structured and Unstructured Codes”, *ASME Turbo Expo*, GT2016-57189.

Invited Seminars:

Sandberg, R.D., 2022, “High-fidelity simulation and machine-learning based modeling of turbomachinery flows”, Seminar at the Propulsion Institute of the University of the Armed Forces, Munich, Germany, 22 July, 2022.

Sandberg, R.D., 2022, “How can high-fidelity simulations combined with machine-learning impact aircraft engine design?”, Seminar at the Propulsion Institute of the University of Braunschweig, Germany, 20 July, 2022.

Sandberg, R.D., 2022, “Scale resolving simulations and machine learning for CFD”, Keynote at the Summer School for Advanced Research in Turbomachinery, Florence, Italy, 28 June, 2022.

Sandberg, R.D., 2021, “Towards exascale simulations for efficient, low-emissions gas turbines - GTx,” PaCER Seminar: Computational Fluid Dynamics - Pawsey Supercomputing Centre, 15 June, 2021.

Sandberg, R.D., 2021, “Gene-expression programming based machine learning for developing turbulence models,” Shear Flow Instability, Transition and Turbulence Seminar Series, Monash University, 28 April, 2021.

Sandberg, R.D., 2021, “How can high-performance computing help design more efficient jet engines: physical insight and machine learning,” Invited talk at the Inaugural National Computing Infrastructure (NCI) presents: TechTake talk, April 27th, 2021.

Sandberg, R.D., 2020, “Combining high-fidelity simulations with machine-learning for physical insight and more accurate low-order models,” Keynote lecture at virtual Global meet on ‘Computational Modeling and Simulation: Recent Innovations, Challenges & Perspectives’ November 4th, 2020.

Sandberg, R.D., 2020, “Combining high-fidelity CFD and Machine-Learning to better predict flow in jet engines,” Invited talk at HPC-AI Advisory Council 2020 Australia Conference, online conference by the HPC-AI Advisory Council in partnership with the National Computing Infrastructure, September 2nd, 2020.

Sandberg, R.D., 2019, “Impact of high-fidelity turbine simulations on design: physical insight and machine learning,” *Invited talk at Future Aviation Workshop*, Korean Advanced Institute of Science and Technology, Daejeon, South Korea, December 5th, 2019.

Sandberg, R.D., 2019, “Impact of high-fidelity turbine simulations on design: physical insight and machine learning,” *Research Seminar Series at Mechanical and Aerospace Engineering Department*, Monash University, October 1st, 2019.

Sandberg, R.D., 2019, “Impact of high-fidelity simulations of turbines: physical insight and machine learning”, *Seminar at Galcit,* California Institute of Technology, USA, 14 June 2019.

Sandberg, R.D., 2019, “Turbomachinery component simulations: challenges, physical insight and modelling”, *Seminar at the School of Aerospace, Mechanical and Mechatronic Engineering*, University of Sydney, New South Wales, 26 March 2019.

Sandberg, R.D., 2018, “Hi-Fi simulations of turbomachinery components: challenges, physical insight and modelling”, *Seminar at the Department of Industrial Engineering*, University of Florence, Florence, 8 June 2018.

Sandberg, R.D., 2018, “Gene expression programming for improving turbulence models”, *Monash eResearch Machine Learning Symposium*, Monash University, Clayton, 19 March 2018.

Sandberg, R.D., 2018, “Simulations of turbomachinery components: challenges, physical insight and modelling”, *Seminar at the Centre for Energy Technology*, University of Adelaide, Adelaide, 2 March 2018.

Sandberg, R.D., 2018, “High-fidelity simulation of turbomachinery components for physical insight and modelling”, *Defense, Science and Technology Group*, Fishermans bend, Melbourne, 1 February 2018.

Sandberg, R.D., Pichler, R., 2017, “Transition mechanisms on High Pressure Turbine Vane midspan sections subject to varying inlet turbulence”, *Invited keynote talk of technical session at 17th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery*, Dec 16-21, 2017, Maui, USA.

Sandberg, R.D., 2017, “DNS/LES of turbomachinery components for physical insight and modelling”, *Invited Keynote at UK turbulence consortium meeting*, Imperial College London, September 4th, 2017.

Sandberg, R.D., 2017, “High-fidelity compressible-flow simulations for physical insight and improving low-order models in turbomachinery”, *Seminar at Hong-Kong University of Science and Technology*, Hong-Kong, March 20th, 2017.

Sandberg, R.D., 2016, “Compressible-Flow DNS and modelling for Turbomachinery”, *Research Seminar at CERFACS*, Toulouse, September 26th, 2016.

Sandberg, R.D., 2016, “Compressible-Flow DNS and modelling for Turbomachinery,” *Research Seminar Series at Mechanical and Aerospace Engineering Department*, Monash University, August 9th, 2016.

Sandberg, R.D., 2016, “Impacting Industry by enabling a step-change in simulation fidelity for flow and noise problems”, *veski fellowship announcement event*, Melbourne, April 28, 2016.

Sandberg, R.D., 2015, “Large-scale Compressible-Flow Direct Numerical Simulations for Turbomachinery,” *Research Seminar Series at School of Mechanical and Manufacturing Engineerin*g, University of New South Wales, August 8th, 2015.

Sandberg, R.D., 2015, “Large-scale Compressible-Flow Direct Numerical Simulations,” Keynote presentation at the *ERCOFTAC Workshop Direct and Large-Eddy Simulations 10*, Limassol, Cyprus, 27-29 May, 2015.

Michelassi, V., 2015, “Unsteady Effects on Turbine Performance by Compressible Direct Numerical and Large Eddy Simulations”, Invited Lecture, 11th European Turbomachinery Conference, March 2015 Madrid.

Michelassi, V., 2015, “Modeling and Resolving Turbulence (and Unsteadiness) in Turbomachinery Flows”, Tutorial at ASME Turbo-Expo Conference, June 2015 Montreal.

Michelassi, V., 2016, “DNS and LES of Realistic Turbomachinery Flows”, Key-Note Lecture, KTH – Linne FLOW Centre, January 2016 Stockholm.

Michelassi, V., “DNS and LES of Turbomachinery Flows: From Research to Design”, Symposium on “The Future of Turbomachinery Research”, Whittle Laboratory, University of Cambridge, January 2018.

Curriculum Vitae

Professor Richard D. Sandberg (PI)
Department of Mechanical Engineering, Level 4, E407
University of Melbourne, Parkville 3010, Australia
+61 383449084
richard.sandberg@unimelb.edu.au

Professional Preparation

2004:	Ph.D. in Aerospace Engineering, Minor in Applied Mathematics, University of Arizona: "*Numerical investigation of transitional and turbulent supersonic axisymmetric wakes*".
1999:	M. S. in Mechanical Engineering, University of Arizona: "*Investigation of Turbulence Models for Turbulent Boundary Layer Flows using Temporal Numerical Simulations*".
1998:	B. S. (equivalent) in Aerospace Engineering, University of Stuttgart, Germany.

Appointments

2022-2022:	Visiting Professor, Dept. Mech. & Industr. Eng., University of Florence
2015–present:	Chair of Computational Mechanics, Dept. Mech Eng., University of Melbourne
2012–2015:	Professor of Fluid Dynamics and Aeroacoustics, Univ. of Southampton
2011–2012:	Senior Lecturer, Univ. of Southampton
2007-2012	Lecturer, Univ. of Southampton
2005-2007:	Postdoctoral Research Fellow, Aerodynamics, Univ. of Southampton

Five Publications Most Relevant to This Proposal

1. **Sandberg, R.D.,** Michelassi, V. 2022, "Fluid Dynamics of Axial Turbomachinery: Blade- and Stage-Level Simulations and Models", *Annual Review of Fluid Mechanics*, 54:255-85.
2. Jelly, T., Nardini, M., Rosenzweig, M., Leggett, J., Marusic, I., **Sandberg, R.D.**, 2022, "High-fidelity computational study of roughness effects on high-pressure turbine performance and heat transfer", proceedings of the *Symposium of Turbulent Shear Flow Phenomena 12* (TSFP-12), Osaka, Japan, July 2022.
3. Zhao, Y., **Sandberg R.D.,** 2021 "High-fidelity Simulations of a High-pressure Turbine Vane: Effect of Exit Mach Number on Losses", *Journal of Turbomachinery*, 143(9): 091002.
4. **Sandberg, R.D.,** Michelassi, V. 2019, "The current state of high-fidelity simulations for main gas path turbomachinery components and their industrial impact", *Flow, Turbulence and Combustion*, pp.1-52.
5. Hammer, F., Sandham, N.D., **Sandberg R.D.**, 2018 "Large Eddy Simulations of a Low-Pressure Turbine Roughness - Modeling and the Effects on Boundary Layer Transition and Losses", ASME IGTI, GT2018-75796.

Research Interests and Expertise

My main interest is in high-fidelity simulation of transitional and turbulent flows, with application to turbomachinery, turbulence model development, aerodynamically generated noise and flow control. In particular, the aim of performing large-scale simulations on the latest HPC computing architectures is to i) gain physical understanding of flow and noise mechanisms and ii) help assess and improve low-order models that can be employed in an industrial context. I was awarded a **veski** innovation fellowship in July 2015 entitled: "Impacting Industry by enabling a step-change in simulation fidelity for flow and noise problems" and more recently an Australian Research Council Future Fellowship (2020-2024) "Unravelling the enigma of turbulence by integrating simulation and data-driven modelling".

I have a broad background in fluid dynamics and aeroacoustics, with specific training and expertise in high-fidelity simulation of fluid flows with application to turbulence, turbomachinery and aeroacoustics. I have extensive experience in conducting large-scale computational fluid dynamics and aeroacoustics simulations, starting in 2001 with an award of a U.S. Department of Defence (DoD) High Performance Computing Challenge project: “High Accuracy DNS and LES of High Reynolds Number, Supersonic Base Flows and Passive Control of the Near Wake”. Following my move from the US to the U.K., I was awarded a Royal Academy of Engineering/EPSRC research fellowship (2007-2012) that enabled me to develop a novel, state-of-the-art compressible flow solver, purposely designed to exploit high-performance computing systems. During that time, I was given early-use access to the U.K.’s national HPC service HECToR (CRAY XE), and then to subsequent new phases of the national services HECToR and ARCHER (CRAY XE and XC). In additional collaborative projects with the CRAY center of excellence, NVIDIA, Oakridge National labs and three GPU Hackathons my team has attended my code was further optimized for performance on HECToR, ARCHER, other CPU-based platforms and the GPU version of the code has been used on the DoE’s TITAN and Summit in previous INCITE and ALCC projects and CSCS’s Piz Daint, and in a project funded by the Australian Pawsey HPC center it is currently being ported to the AMD CPU/GPU architecture that will be used for Frontier.

As PI or co-Investigator on several industry- and Australian Research Council and UK Engineering and Physical Sciences Research Council-funded grants, I laid the groundwork for another important part of the proposed research by developing machine learning algorithms for translating high-fidelity simulation data to low-order models. Since my move to Australia in 2015, I have further optimized my DNS/LES code to perform well on the latest NVIDIA V100 architecture on IBM (Minsky) systems and more recently also for NVIDIA A100 and AMD GPU architectures.

Using data generated using high-performance computing I have published more than 290 articles, with 110 of them in leading journals of the research fields relevant to this proposal, e.g. *J. Fluid Mech*., *J. Turbomachinery, J. Comp. Physics, J. Sound and Vibr., J. Applied Mech.*

Synergistic Activities

1. Member of ASME’s Gas Turbine Technology Group (GTTG) leadership team
2. Executive committee member of the Melbourne Energy Institute, Program lead for Power Generation and Transport
3. Development of High-Performance DNS/LES code that is now used by several groups in Australia, Europe and North America
4. PI on Australian Research Council Linkage Infrastructure, Equipment and Facilities Grant, LE170100200, “A high-performance cloud resource for computational modelling”.
5. Coordinator of the UK Turbulence Consortium (2011-2015)
6. Member of Australian National Computational Merit Allocation Committee (2016-2019)

Collaborators (*past 5 years including name and current institution*)

Dr. John Eaton, Stanford University
Dr. Nicholas Hutchins, University of Melbourne
Dr. Markus Klein, University of the Armed Forces Munich
Dr. Greg Laskowski, Dassault Systemes
Dr. Ivan Marusic, University of Melbourne
Dr. Beverley McKeon, CalTech
Dr. Vittorio Michelassi, Baker Hughes, Florence
Dr. Stephane Moreau, University of Sherbrooke
Dr. Nagabhushana Vadlamani Rao, Indian Institute of Technology Madras
Dr. Roberto Pacciani, University of Florence
Dr. Edward Richardson, University of Southampton
Dr. Neil Sandham, University of Southampton
Dr. Dong Hyuk Shin, Korean Advanced Institute of Science and Technology
Dr. Koichi Tanimoto, Mitsubishi Heavy Industries
Dr. Andrew Wheeler, Cambridge University

Professor Ivan Marusic FAA FTSE
Department of Mechanical Engineering
The University of Melbourne, Victoria 3010 Australia
Tel: +61 3 8344 6872; Mobile: +61 421 996 024
Email: imarusic@unimelb.edu.au

Professional Preparation

1987	B.E.(Hons)Mech, University of Melbourne
1992	Ph.D. University of Melbourne

Appointments

2017-2021	Deputy Dean (Research), Faculty of Engineering & IT, Univ. of Melbourne
2016-	Redmond Barry Distinguished Professor, Univ. of Melbourne
2006-2016	Professor, Dept. Mechanical Engineering, Univ. of Melbourne
2015	Visiting Professor, Princeton University (Sep-Dec), Johns Hopkins University (Oct)
2006	Professor, Dept. Aerospace Engineering and Mechanics, Univ. of Minnesota
2002-2006	Associate Professor, Dept. Aerospace Engineering and Mechanics, Univ. of Minnesota
2001-2002	Visiting Associate Professor, GALCIT, California Institute of Technology
1998-2002	Assistant Professor, Dept. Aerospace Engineering and Mechanics, Univ. of Minnesota
1993-1998	Senior Research Fellow, Department of Mechanical Engineering, Univ. of Melbourne
1992	Research Fellow, Department of Mechanical Engineering, Univ. of Melbourne

Five Publications Most Relevant to This Proposal

1. Hutchins, N. & **Marusic, I.** (2007) Evidence of very long meandering structures in the logarithmic region of turbulent boundary layers. *J. Fluid Mech*. 579, 1-28.
2. Mathis, R., Hutchins, N. & **Marusic, I.** (2009) Large-scale amplitude modulation of the small-scale structures of turbulent boundary layers. *J. Fluid Mech.* 628, 311-337.
3. **Marusic, I.**, Mathis, R. & Hutchins, N. (2010) Predictive model for wall-bounded turbulent flow. *Science* 329: 193-196.
4. **Marusic, I.**, McKeon, B.J., Monkewitz, P.A, Nagib, H.M., Smits, A.J. & Sreenivasan, K.R. (2010) Wall-bounded turbulent flows at high Reynolds numbers: Recent advances and key issues. *Phys. Fluids* 22, 065103, 1-24.
5. **Marusic, I.** & Monty, J.P. (2019) Attached eddy model of wall turbulence. *Annual Rev. Fluid Mech.,* 51, 49-74.

Research Interests and Expertise

Professor Marusic is well-recognised internationally as a leading researcher in the area of experimental fluid mechanics and turbulent flows. He has pioneered the development of novel analytical tools, experimental methods and instrumentation, which have produced unique datasets vital for our understanding of complex flows. This includes developments in micro-particle image velocimetry. He is best known for the Marusic & Perry wall-wake attached eddy model for turbulent flows, a physical model that provides a conceptual framework with which to consider the complex flow physics. In 2007, Hutchins and Marusic reported the discovery of 'superstructures' in turbulent flows, very-large scale organised motions that have been shown to be the dominant contributor to momentum transport in all manifestations of wall turbulence. Since their discovery, superstructures have been the focus of intense study by various groups around the world. In total, Marusic has published 201 journal articles to date. This includes 78 papers in the *Journal of Fluid Mechanics*, which is regarded as the top journal in the field, two in *Science*, three in *Physical Review Letters*, 34 in *Physics of Fluids* or *Physical Review Fluids*, and 19 in *Experiments in Fluids*. He has 49 papers with more than 100 citations, 12 with >300 citations. His h-index is 68 (Google Scholar), 55 (Web of Science) and 58 (Scopus). His contributions have been recognised through a number of prestigious awards with the Woodward Medal at the University of

Melbourne and the Stanley Corrsin Award from the American Physical Society (APS). He is a Fellow of the APS, the Australasian Fluid Mechanics Society and the Australian Academy of technology and Engineering. In 2014 he was elected as a Fellow of the Australian Academy of Science. He has held a number of prestigious fellowships, including an ARC Laureate Fellowship (2012-2017), ARC Federation Fellowship (2006-2011), and a Packard Fellowship in Science and Engineering (2001-2006). He has been an Associate Editor for the *Journal of Fluid Mechanics* since 2008, and was the augural President and Founding member of the Australasian Fluid Mechanics Society.

Synergistic Activities

<u>Society Memberships</u>
Fellow, Australian Academy of Technology and Engineering (elected 2021)
Fellow, Australian Academy of Science (elected 2014)
Fellow, Australasian Fluid Mechanics Society (elected 2014)
Fellow, American Physical Society (elected 2010)
Senior Member, American Institute of Aeronautics and Astronautics
<u>Editorships</u>
Associate Editor (2008-current), *Journal of Fluid Mechanics*
Associate Editor (2011-2021), *Journal of Hydraulic Research*
Editor (2007-2012), *Experimental Thermal Fluid Science*
Editorial Board (2007-2021), IOP *Measurement Science and Technology*
<u>Professional Service</u>
Elected, Council Ordinary Member (Physical Sciences), Australian Academy of Science (2019-)
Chair of Advisory Committee, Max Planck - University of Twente Center for Complex Fluid Dynamics
Treasurer, Australasian Fluid Mechanics Society (2012-2021)
Inaugural President, Australasian Fluid Mechanics Society (2010-2012)
Member & Chair, National Committee for Mech. Sciences, Australian Academy of Science (2010-2015)
Nominating Committee (2012-2014), American Physical Society, Division of Fluid Dynamics
College of Experts (2009-2011), Australian Research Council
Chair (2011), Engineering, Mathematics and Informatics Panel, Australian Research Council
Head of Organizing Committee, Turbulence and Shear Flow Phenomena (TSFP9), Melbourne 2015

Collaborators (*past 5 years including name and current institution*)

A/Prof Daniel Chung – University of Melbourne
Prof Bharathram Ganapathisubramani – University of Southampton
A/Prof Michele Guala – University of Minnesota
Prof Marcus Hultmark – Princeton University
Prof Nicholas Hutchins – University of Melbourne
Dr Simon Illingworth – University of Melbourne
Prof Mihailo Jovanovic - University of Southern California
Prof Christian Kähler - Universität der Bundeswehr München
Prof Joseph Klewicki – University of Melbourne
Prof Detlef Lohse – University of Twente
Prof Charles Meneveau – Johns Hopkins University
Prof Jason Monty – University of Melbourne
Prof Vladimir Nikora – University of Aberdeen
Prof Andrew Ooi – University of Melbourne
Dr Jimmy Philip – University of Melbourne
Prof Maarten van Reeuwijk - Imperial College London
Prof Richard D Sandberg - University of Melbourne
Prof Alexander Smits – Princeton University
Prof Alessandro Talamelli – University of Bologna

Sriram Shankaran
Consulting Engineer
GE Aviation
T: +1 (518) 763-7682
E: shankaran@ge.com

Professional Preparation

2003 PhD Aeronautics and Astronautics, Stanford University, CA
1998 MS Aerospace Engineering, IIT Madras, India
1996 BTech Aerospace Engineering, IIT Madras, India

Appointments

2018-Present	Consulting Engineer, Aerothermal Division, GE Aviation, Cincinnati OH
2015-2018	Sub-Section manager for Aero-Thermal Methods, GE Aviation, Cincinnati OH
2012-2015	Manager for Fluid Mechanics Lab, GE Research Center, Niskayuna, NY
2007-2012	Mechanical Engineer, Fluid Mechanics Lab, GE Research Center, Niskayuna, NY

Five Publications Most Relevant to This Proposal

1. **Sriram Shankaran** and Andre Marta, Improving the performance of Turbomachinery components through Sensitivity of Boundary Conditions using Adjoints and Control Theory, *AIAA Journal of Propulsion and Power*, 2014.
2. **Sriram Shankaran**, Andre Marta, Prem Venugopal, Qiqi Wang, Interpretation of Adjoint Solutions for Turbomachinery Flows, *ASME Turbo Expo 2012*, Copenhagen, Denmark, *AIAA Journal*, 2013.
3. Andre Marta and **Sriram Shankaran**, On the use of Constant-Eddy Viscosity Adjoints Models for Turbomachinery Design, *Computers and Fluids*, 2012.
4. **Sriram Shankaran** and Antony Jameson, Robust Optimal Control using Polynomial Chaos and Adjoints for Systems with Uncertain Inputs, AIAA Fluids Conference, Hawaii, 2011, *AIAA Journal*, 2012.
5. **Sriram Shankaran** and Brian Barr, Efficient Gradient-Based Algorithms for the Construction of Pareto Fronts, ASME Turbo Expo 2011, *Journal of Optimization Engineering*, 2011.

Research Interests and Expertise

- Computational Methods
- Aerodynamics
- Computational Fluid Dynamics

Synergistic Activities

Reviewer for:

- ASME Turbo Expo
- AIAA Journal
- Journal of Turbomachinery
- Journal of Propulsion and Power

Collaborators (Past 5 Years)
Prof. Richard Sandberg, University of Melbourne, Australia
Prof. Qiqi Wang, MIT, MA
Prof. Andre Marta, University of Lisbon, Portugal
Prof. Antony Jameson, Stanford/Texas A&M
Prof. Claire Tomlin, University of Berkeley, CA

Dr. Aamir Shabbir
Principal Engineer
GE Aviation, 1 Neumann Way
Cincinnati, OH 4521, USA
aamir.shabbir@ge.com

Professional Preparation

- Ph.D. - Mechanical Engineering, 01/1987
  State University of New York, Buffalo, NY 14260, USA
- M.S. - Mechanical Engineering, 05/1983
  State University of New York, Buffalo, NY 14260, USA
- B.S. - Mechanical Engineering, 05/1981
  University of Engineering and Technology, Lahore, Pakistan

Appointments

2013 to Present, Principal Engineer
GE Aviation, 1 Neumann Way, Cincinnati, OH 45215
CFD, LES, aerodynamics of turbomachinery

2008 to 2013, Staff Research Engineer
Pratt & Whitney and United Technologies Research Center
United Technologies Corporation, CT 06108
Tackled problems associated with the aerodynamic performance of turbomachinery using computational and analytical methods, and established guidelines for standard work.

1989 to 2008, Sr. Research Associate
NASA Glenn Research Center, Cleveland OH 44139
Advanced the state-of-the-art of CFD tools for propulsion systems by implementing advanced models into the Computational Fluid Dynamics (CFD) codes, developing associated software tools, simulating a variety of propulsion system geometries.

Five Publications Most Relevant to This Proposal

1. Loss Analysis of Unsteady Turbomachinery Flows Based on the Mechanical Work Potential, J. Leggett, E. Richardson, S. Priebe, **A. Shabbir**, V. Michelassi, & R. Sandberg, J. Turbomach., 2020, 142(11): 111009
2. Loss Prediction in an Axial Compressor Cascade at Off- Design Incidences With Free Stream Disturbances Using Large Eddy Simulation, J. Leggett, S. Priebe, **A. Shabbir**, R. Sandberg, E. Richardson, & V. Michelassi, J. Turbomach. 2018, Vol. 140, 071005-1 - 071005-11.
3. Aerodynamic Analysis of a Boundary-Layer-Ingesting Distortion-Tolerant Fan. R.V. Florea, D. Voytoovych, G. Tillman, M. Stucky, **A. Shabbir**, O.Sharma, D.J. Arend. Paper GT2013-94656, ASME TURBO EXPO 2013.
4. Parametric Analysis and Design for Embedded Engine Inlets. R. Florea, C. Matalanis, L.W. Hardin, M. Stucky, **A. Shabbir**. 48th AIAA/ASME/SAE/ASEE Joint Propulsion Confernce and Exhibit, 2012.
5. A New K-Epsilon Eddy Viscosity Model for High Reynolds Number Turbulent Flows. T.-H. Shih, W.W. Liou, **A. Shabbir**, Z. Yang and J. Zhu. Computers Fluids, 1995, Vol. 24, No. 3, pp. 227-23.

Research Interests and Expertise

I have worked in the area of CFD of turbomachinery and turbulence modeling for 28 years during which time my responsibilities have included; developing turbulence models for turbomachinery applications, developing CFD codes, establishing CFD best practices, and using CFD to understand turbomachinery flow physics. I have supported the design of fans, boosters, and compressors during my tenure at the engine companies. Was member of the team that won the NASA Group Achievement Award in 2017 for the design of boundary layer ingesting fan stage. I was co-winner of the Best Paper of the Year award from the American Society of Mechanical Engineers in 2005 for the paper that explains the flow mechanism responsible for the compressor stall benefit with the use of casing treatment. I was a co-developer of one of the turbulence models that is available in the commercial CFD code FLUENT. Recently I have been actively involved in the use of LES to understand the key physics of turbomachinery aerodynamics by using simplified geometries. In this regard I have worked closely with the University of Melbourne and the University of Southampton.

Synergistic Activities

1. LES of compressor airfoils to understand loss as a function of flow incidence
2. LES of compressor airfoils to understand the loss from unsteadiness due to rotor-stator interaction
3. Unsteady RANS of multistage turbomachinery

Collaborators (*past 5 years including name and current institution*)

Dr. Stephan Priebe, GE Global Research Center, Niskayuna, NY, USA
Dr. John Leggett, University of Melbourne, Melbourne, Australia
Dr. Vittorio Michelassi, Baker Hughes, Florence, Italy
Prof. Richard Sandberg, University of Melbourne, Melbourne, Australia
Prof. Edward Richardson, University of Southampton, Southampton, UK

Dr. William Solomon
Chief Consulting Engineer for Compression Aerodynamics
GE Aviation, 1 Neumann Way
Cincinnati, OH 4521, USA
William.j.solomon@ge.com

Professional Preparation

- Ph.D. - "Unsteady Boundary Layer Transition on Axial Compressor Blades", 03/1997
  University of Tasmania, Australia
- B.E. Hons - Mechanical/Electrical Engineering, 1991
  University of Tasmania, Australia

Appointments

2021 – Present: Chief Consulting Engineer for Compression Aerodynamics, GE Aviation
2019 – 2021: Consulting Engineer, GE Aviation
2016 – Present: Principal Engineer, Fan and Compressor Aerodynamics Design, GE Aviation
2012 – 2016: Manager, Fan and Compressor Aerodynamics Sub-Section, GE Aviation
2006 – 2012: Senior Engineer, Fan and Compressor Aerodynamics Design, GE Aviation
2000 – 2006: Lead Engineer, Fan and Compressor Aerodynamics Design Group, GE Aviation
1997 – 2000: Senior Research Associate, Ohio Aerospace Institute
1996 – 1997: Development Engineer, GEC Alsthom Turbine Generators Limited, Rugby, UK

Five Publications Most Relevant to This Proposal

1. Shin, H-W, **Solomon, W. J.** and Wadia, A. 2008, Transonic Fan Tip-Flow Features Revealed by High Frequency Response Over-tip Pressure Measurements, ASME Paper GT2008-50279
2. **Solomon, W. J.**, 2000, Effects of Turbulence and Solidity on the Boundary Layer Development in a Low Pressure Turbine, ASME Paper 2000-GT-0273
3. Walker, G. J., Hughes, J. D. and **Solomon, W. J.**, 1998, Periodic transition on an axial compressor stator - Incidence and clocking effects. Part I - Experimental data, ASME Paper 98-GT-363
4. **Solomon, W. J.**, Walker, G. J. and Hughes, J. D., 1998, Periodic transition on an axial compressor stator - Incidence and clocking effects. Part II - Transition onset predictions, ASME Paper 98-GT-364
5. **Solomon, W. J.**, Walker, G. J. and Gostelow, J. P., 1996, Transition length prediction for flows with rapidly changing pressure gradients, ASME Journal of Turbomachinery, 118, no. 4, 744-751, (ASME Paper 95-GT-241)

Research Interests and Expertise

Research activities in the early part of my career focused on the problem of understanding boundary-layer transition evolution and mechanisms in turbomachinery flows. The flows in these machines are inherently unsteady. Understanding of boundary-layer transition physics and of the post-processing techniques that were applied to interpret experimental data from hot wire and surface hot film in compressors and turbines are finding new relevance as the computational techniques such as LES and DNS are catching up with the full level of physical complexity.

My subsequent career at GE has greatly expanded my understanding of the full range of turbomachinery design challenges and given me an opportunity to support and lead the development of a diverse range of products. Of particular relevance to the current study, I have completed a GE review of the impact of airfoil surface roughness on key compressor product performance data with comparison to proprietary modeling.

In addition to the design experience, collaboration with other workers at GE Aviation, GE Global Research and the University of Melbourne on the problem of improving our design capability has given me the opportunity to participate in and guide the application of LES towards the applications which are most likely to be impactful in improving our designs to support the overall goal of reducing fuel burn and CO2 emissions from our next generation of products. Most recently this includes collaboration on an upcoming paper "LARGE EDDY SIMULATION OF LAMINAR AND TURBULENT SHOCK/BOUNDARY LAYER INTERACTIONS IN A TRANSONIC PASSAGE", Priebe et at, originally planned for release at the 2020 ASME Turbo Expo.

Collaborators (*past 5 years including name and current institution*)

Dr. Stephan Priebe, GE Global Research Center, Niskayuna, NY, USA
Mr Daniel Wilkin, GE Aviation, Evendale, OH, USA
Mr Andy Breez-Stringfellow, GE Aviation, Evendale, OH, USA
Dr. Vittorio Michelassi, Baker Hughes, Florence, Italy
Dr. Richard Sandberg, University of Melbourne, Melbourne, Australia
Dr Ken Suder, NASA GRC, USA

Paul Vitt
Chief Consulting Engineer for Turbine Aerodynamics
General Electric Company - Aviation
M: +1 203 727 3543
paul.vitt@ge.com

Professional Preparation

1992 MSc Mechanical & Aerospace Engineering, University of Missouri-Rolla, Rolla, MO
1991 BSc Mechanical & Aerospace Engineering, University of Missouri-Rolla, Rolla, MO

Professional Experience

2020-Present Chief Consulting Engineer for Turbine Aerodynamics, GE Aviation, Cincinnati OH
2019-2020 Consulting Engineer - Turbine Aerodynamics, GE Aviation, Cincinnati OH
2013-2019 Sub-Section Manager - Turbine Aerodynamics, GE Aviation, Cincinnati OH
2010-2013 Center Manager, QuEST Global Services, Cincinnati, OH
2008-2010 Program Manager, QuEST Global Services, Cincinnati, OH

Publications Most Relevant to This Proposal

1. **Vitt, P.**, Iverson, C., Malak, M., Liu, J.: Impact of Flowfield Unsteadiness on Film Cooling of a High Pressure Turbine Blade," ASME Paper GT2010-22773, presented at ASME Turbo Expo 2010, Glasgow, Scotland, June 2010.
2. Molter, S., Dunn, M., Haldeman, C., Bergholz, R., **Vitt, P.**: "Heat-Flux Measurements and Predictions for the Blade Tip Region of a High-Pressure Turbine," GT2006-9004, 2006.
3. Cherry, D., Wadia, A., Beacock, R., Subramanian, M., **Vitt, P.**: "Analytical investigation of a low pressure turbine with and without flowpath endwall gaps, seals, and clearance features," ASME paper 2005-68492, 2005.
4. Barter, J., **Vitt, P.**, Chen, J.P.: "Interaction Effects in a Transonic Turbine Stage," ASME paper 2000-GT-0376, 2000.

Research Interests and Expertise

- Aerodynamics and design of aero components
- Computational Fluid Dynamics
- Testing and validation

Synergistic Activities

- Development of advanced design and analysis capabilities within GE Aviation
- Test validation activities and proposals for Aviation engines
- Integration of advanced methods into design applications for Aviation engines

Dr. John Leggett
Department of Mechanical Engineering
University of Melbourne, Parkville 3010, Australia
jake.leggett@unimelb.edu.au

Professional Preparation

2018: PhD at the University of Southampton, UK -Focused on Large Eddy Simulations of Axial compressor cascades
2012: MSc in Advanced Computational Methods for Fluid Structural Interaction at Imperial College London, UK
2009: MEng in Mechanical engineering at the University of Manchester, UK

Appointments

2019 March – present: Research Fellow at the University of Melbourne

Five Publications Most Relevant to This Proposal

1. Jelly, T., Nardini, M., Rosenzweig, M., **Leggett, J.**, Marusic, I., Sandberg, R.D., 2022, "High-fidelity computational study of roughness effects on high-pressure turbine performance and heat transfer", proceedings of the *Symposium of Turbulent Shear Flow Phenomena 12* (TSFP-12), Osaka, Japan, July 2022.
2. Zhao, Y., **Leggett, J.**, Sandberg R.D., 2022 "High-Fidelity Simulation Study of the Unsteady Flow Effects on High-Pressure Turbine Blade Performance", Proceedings of *ASME IGTI TurboExpo*, GT2022-82370.
3. Przytarski, P. J., **Leggett, J.**, Sandberg R.D., Bode, C., 2022 "Highly Resolved Large-Eddy Simulations of a Transonic Compressor Stage Midspan Section - Part II: Effect of Rotor-Stator Gap", Proceedings of *ASME IGTI TurboExpo*, GT2022-82474.
4. **Leggett, J.**, Richardson, E., Priebe, S., Shabbir, A., Michelassi, V., Sandberg R.D., 2020 "Loss Analysis of Unsteady Turbomachinery Flows Based on the Mechanical Work Potential", *Journal of Turbomachinery*, 142(11): 111009.
5. **Leggett, J.**, Sandberg R.D., Priebe, S., Richardson, E., Shabbir, A., Michelassi, V., 2018 "Loss Prediction in an Axial Compressor Cascade at Off-Design Incidences with Free Stream Disturbances", *Journal of Turbomachinery* 140(7), pp. 071005

Research Interests and Expertise

Research interests are in high fidelity numerical simulation of gas turbine machinery. Focused on Large Eddy Simulations and loss model development. Expertise are in High Performance Computing, Large Eddy Simulations, Data Management and loss analysis.

Collaborators (*past 5 years including name and current institution*)

Dr. Aamir Shabbir, GE Aviation, Cincinnati, OH, USA
Dr. Edward Richardson, University of Southampton, Southampton
Dr. Richard Sandberg, University of Melbourne, Melbourne
Dr. Stephan Priebe GE Global Research, Niskayuna, NY, USA
Dr. Vittorio Michelassi Baker Hughes, Florence Italy

Dr. Pawel Jan Przytarski
Department of Mechanical Engineering
University of Melbourne, Parkville 3010, Australia
+44 7729881854
pawel.przytarski@unimelb.edu.au

Professional Preparation

2020: Ph.D.: "*High Fidelity Simulation of Loss Mechanisms in Compressors",* Whittle Laboratory, University of Cambridge, UK
2016: MRes: *"High Fidelity Simulation for Turbomachinery"*, Whittle Laboratory, University of Cambridge, UK
2015: MEng in Mechanical Engineering, Imperial College London, UK

Appointments

2020–present: Postdoctoral Research Fellow, Dept. Mech Eng., University of Melbourne.

Five Publications Most Relevant to This Proposal

1. Bode, C., **Przytarski, P. J.**, Leggett, J., Sandberg R.D., 2022 "Highly Resolved Large-Eddy Simulations of a Transonic Compressor Stage Midspan Section - Part I: Effect of Inflow Disturbances", Proceedings of *ASME IGTI TurboExpo*, GT2022-81673.
2. **Przytarski, P. J.**, Leggett, J., Sandberg R.D., Bode, C., 2022 "Highly Resolved Large-Eddy Simulations of a Transonic Compressor Stage Midspan Section - Part II: Effect of Rotor-Stator Gap", Proceedings of *ASME IGTI TurboExpo*, GT2022-82474.
3. **Przytarski, P. J.,** Lengani, D., Wheeler, A. P. S., Simoni, D., "Data-driven analysis of high fidelity simulation of multi-stage compressor", Proceedings of XIV European Conference on Turbomachinery Fluid dynamics and Thermodynamics, ETC14, Gdansk, Poland
4. **Przytarski, P. J.** and Wheeler, A. P. S., "The effect of gapping on compressor performance", J. Turbomachinery, Dec 2020, 142(12)
5. **Przytarski, P. J.** and Wheeler, A. P. S., "Accurate prediction of loss using high fidelity methods", J. Turbomachinery, Mar 2021, 143(3)
6. Spencer, R., **Przytarski, P. J.**, Adami, P., Grothe, P. and Wheeler, A. P. S., "Importance of non-equilibrium modelling for compressors", Proceedings of *ASME IGTI TurboExpo*, Virtual, Online

Research Interests and Expertise

My research interests lie in high-fidelity simulations of turbomachinery flows and how high-fidelity datasets can be harnessed to improve our understanding of flow physics and inform industrial design; in particular I am interested in data-driven approaches to flow decomposition and data reduction. I gained extensive experience performing large scale HPC simulations of turbomachinery flows, developing in-house solvers and processing tools as well as analysing large datasets.

Collaborators (*past 5 years including name and current institution*)

Prof. Richard Sandberg, University of Melbourne, Australia
Dr Andrew Wheeler, University of Cambridge
Dr. Davide Lengani, University of Genoa, Italy
Dr. Daniele Simoni, University of Genoa, Italy
Prof. Luca di Mare, University of Oxford, UK

Dr. Melissa Kozul
Department of Mechanical Engineering
University of Melbourne, Parkville 3010, Australia
+61 421 941 387
kozulm@unimelb.edu.au

Professional Preparation

2018: Ph.D. in Mechanical Engineering, University of Melbourne:
'*The turbulent boundary layer studied using novel numerical frameworks'*
2013: B.E.(Hons)Mech, B.Sc. (Mathematical Physics) University of Melbourne

Appointments

2021 – Present: Postdoctoral Research Fellow, Dept. Mech Eng., University of Melbourne
2018 – 2021: Postdoctoral Fellow, Dept. Energy and Process Eng., NTNU, Trondheim, Norway

Five Publications Most Relevant to This Proposal

1. **Kozul, M.**, Costa, P. S., Dawson, J. R. & Brandt, L., 2020 Aerodynamically driven rupture of a liquid film by turbulent shear flow, *Phys. Rev. Fluids*, Vol. 5, 124302
2. **Kozul, M.**, Hearst, R. J., Monty, J. P., Ganapathisubramani, B. & Chung, D. 2020 Response of the temporal turbulent boundary layer to decaying free-stream turbulence, *J. Fluid Mech*., Vol. 896, A11
3. **Kozul, M.**, Koothur, V., Worth, N. A. & Dawson, J. R. 2019 A scanning particle tracking velocimetry technique for high-Reynolds number turbulent flows, *Exp. Fluids*, 60:137
4. **Kozul, M.**, Chung, D. & Monty, J. P. 2016 Direct numerical simulation of the incompressible temporally developing turbulent boundary layer, *J. Fluid Mech*., Vol. 796, pp. 437-472

Research Interests and Expertise

My research expertise is in the high-fidelity simulations of fundamental turbulent flows that feature critically in energy and transport technologies, for which I often employ strategically-designed numerical 'thought experiments' to decouple effects that are conflated, or add controlled physics. To date my research interests have included wall-bounded flows, homogeneous isotropic turbulence, turbulent multiphase flows as well as particle tracking and sizing algorithms for use with laboratory turbulent flows. I currently lead the 2022 INCITE project "DNS of Performance-enhancing Riblets on Gas Turbine Compressor Blades" (ENG112), which involves collaboration with our project partners at GE.

Collaborators (*past 5 years including name and current institution*)

Prof. Richard Sandberg, University of Melbourne, Australia
Prof. James R. Dawson, NTNU, Norway
Prof. Luca Brandt, KTH, Sweden
Prof. Tamer A. Zaki, Johns Hopkins University, United States of America
A/Prof Daniel Chung, University of Melbourne
Prof Jason Monty, University of Melbourne
Prof. Bharathram Ganapathisubramani, University of Southampton, UK
Dr. Andrea Gruber, SINTEF Energy Research, Trondheim, Norway

Dr. Massimiliano Nardini
Department of Mechanical Engineering
University of Melbourne, Parkville 3010, Australia
+61 (0) 410 873 557
nardinim@unimelb.edu.au

Professional Preparation

2019: Ph.D. in Mechanical Engineering, University of Melbourne:
'*Reduced-order modeling for membrane wings at low Reynolds numbers'*
2014: M.Sc. Aerospace Engineering, University of Pisa
2013: Visiting scholar, San Diego State University
'Computational Multi-purpose Aerodynamics Solver for the Analysis of Innovative Wing Configurations, Helicopter Blades and Wind Turbines'
2011: B.Sc. Aerospace Engineering, University of Pisa

Appointments

2019 – Present: Postdoctoral Research Fellow, Dept. Mech Eng., University of Melbourne

Five Publications Most Relevant to This Proposal

1. Ananth, S.M., **Nardini, M.**, Vaid, A., Vadlamani, N.R., Sandberg, R.D., 2022 "Profile Loss Reduction of High Lift Turbine Blades With Rough and Ribbed Surfaces", *ASME IGTI*, GT2022-82558.
2. Malathi, A. S., **Nardini, M.**, Vaid, A., Vadlamani, N. R., and Sandberg, R. D. "On the efficacy of riblets toward drag reduction of transitional and turbulent boundary layers", AIAA 2022-0472. *AIAA SCITECH 2022 Forum*. January 2022.
3. Jelly, T. O., **Nardini, M.**, Rosenzweig, M., Leggett, J., Marusic, I., Sandberg, R. D. 2022 "High-fidelity computational study of roughness effects in high pressure turbine performance and heat transfer", *12th International Symposium on Turbulence and Shear Flow Phenomena (TSFP12), Osaka, Japan, July 19-22*

Research Interests and Expertise

My main research interest is in computational fluid dynamics, with a focus on high-fidelity simulations of compressible turbulent flows applied to aeroacoustics, turbomachinery and fluid-structure interaction. My areas of expertise include immersed boundary methods and in the past years I worked on developing an efficient, unsteady three-dimensional Boundary Data Immersion Method (BDIM) to simulate complex moving and non-moving geometries immersed in a flow. My code developments have been successfully employed to simulate small-scale surface roughness on high-pressure turbine blades as part of our 2021 INCITE project, and engineered micro-grooves (riblets) on high-pressure compressor blades as part of our current 2022 INCITE (ENG112) project. The data extracted from these large-scale, first-of-a-kind high-fidelity simulations will provide an understanding of the fundamental mechanisms that affect skin friction and heat transfer in the presence of surface imperfections, a condition that typically occurs on real-life blades due to in-service wear and pitting.

Collaborators (*past 5 years including name and current institution*)

Prof. Richard Sandberg, University of Melbourne,

Dr. Simon J. Illingworth, University of Melbourne
Dr. Nagabhushana Vadlamani Rao, Indian Institute of Technology Madras
Dr. Thomas Oliver Jelly, University of Leicester
Prof. Ivan Marusic, University of Melbourne

Section 6: Software Applications and Packages

Question #1

Please list any software packages used by the project, and indicate if they are on open source or export controlled.

Application Packages

Package Name

HiPSTAR (not full Open Source but available to others subject to user agreement)

Indicate whether Open Source or Export Controlled.

Open Source

Section 7: Wrap-Up Questions

Question #1

National Security Decision Directive (NSDD) 189 defines Fundamental Research as "basic and applied research in science and engineering, the results of which ordinarily are published and shared broadly within the scientific community, as distinguished from proprietary research and from industrial development, design, production, and product utilization, the results of which ordinarily are restricted for proprietary or national security reasons." Publicly Available Information is defined as information obtainable free of charge (other than minor shipping or copying fees) and without restriction, which is available via the internet, journal publications, textbooks, articles, newspapers, magazines, etc.

The INCITE program distinguishes between the generation of proprietary information (deemed a proprietary project) and the use of proprietary information as input. In the latter, the project may be considered as Fundamental Research or nonproprietary under the terms of the nonproprietary user agreement. Proprietary information, including computer codes and data, brought into the LCF for use by the project - but not for generation of new intellectual property, etc., using the facility resources - may be protected under a nonproprietary user agreement.

Proprietary Information

Are the proposed project and its intended outcome considered Fundamental Research or Publicly Available Information?

Yes

Will the proposed project use proprietary information, intellectual property, or licensing?

No

Will the proposed project generate proprietary information, intellectual property, or licensing as the result of the work being proposed?

If the response is Yes, please contact the INCITE manager, INCITE@doeleadershipcomputing.org, prior to submittal to discuss the INCITE policy on proprietary work.

No

Question #2

The following questions are provided to determine whether research associated with an INCITE proposal may be export controlled. Responding to these questions can facilitate - but not substitute for - any export control review required for this proposal.

PIs are responsible for knowing whether their project uses or generates sensitive or restricted information. Department of Energy systems contain only data related to scientific research and do not contain personally identifiable information. Therefore, you should answer "Yes" if your project uses or generates data that fall under the Privacy Act of 1974 U.S.C. 552a. Use of high-performance computing resources to store, manipulate, or remotely access any national security information is prohibited. This includes, but is not limited to, classified information, unclassified controlled nuclear information (UCNI); naval nuclear propulsion information (NNPI); and the design or development of nuclear, biological, or chemical weapons or of any weapons of mass destruction. For more information contact the Office of Domestic and International Energy Policy, Department of Energy, Washington DC 20585, 202-586-9211.

Export Control

Does this project use or generate sensitive or restricted information?

No

Does the proposed project involve any of the following areas?

i. Military, space craft, satellites, missiles, and associated hardware, software or technical data

ii. Nuclear reactors and components, nuclear material enrichment equipment, components (Trigger List) and associated hardware, software or technical data

iii. Encryption above 128 bit software (source and object code)

iv. Weapons of mass destruction or their precursors (nuclear, chemical and biological)

No

Does the proposed project involve International Traffic in Arms Regulations (ITAR)?

No

Question #3

The following questions deal with health data. PIs are responsible for knowing if their project uses any health data and if that data is protected. Note that certain health data may fall both within these questions as well as be considered sensitive as per question #2. Questions regarding these answers to these questions should be directed to the centers or program manager prior to submission.

Health Data

Will this project use health data?

No

Will this project use human health data?

No

Will this project use Protected Health Information (PHI)?

No

Question #4

The PI and designated Project Manager agree to the following:

Monitor Agreement

I certify that the information provided herein contains no proprietary or export control material and is correct to the best of my knowledge.

Yes

I agree to provide periodic updates of research accomplishments and to

acknowledge INCITE and the LCF in publications resulting from an INCITE award.

Yes

I agree to monitor the usage associated with an INCITE award to ensure that usage is only for the project being described herein and that all U. S. Export Controls are complied with.

Yes

I understand that the INCITE program reserves the right to periodically redistribute allocations from underutilized projects.

Yes

Section 8: Outreach and Suggested Reviewers

Question #1

By what sources (colleagues, web sites, email notices, other) have you heard about the INCITE program? This information will help refine our outreach efforts.

Outreach

By what sources (colleagues, web sites, email notices, other) have you heard about the INCITE program? This information will help refine our outreach efforts.

We currently have an INCITE2022 award and are on the email distribution list.

Question #2

Suggested Reviewers

Suggest names of individuals who would be particularly suited to assess the proposed research.

Prof. Jeffrey Bons, Ohio State, email: bons.2@osu.edu (expertise with roughness in gas turbines - experimental)

Prof. Krishnan Mahesh, U Minnesota, email: kmahesh@umn.edu (expertise with simulation of rough-wall flows)

Prof. Charles Meneveau, Johns Hopkins, email: meneveau@jhu.edu (expertise with simulation of rough-wall flows)

Prof. Zoltán S. Spakovszky, MIT, email: zolti@mit.edu (expertise with gas turbines)

Dr. Ricardo Garcia-Mayoral, Cambridge, email: rg471@eng.cam.ac.uk (focuses on roughness effects and drag reduction in turbulent flows - modeling and simulation)

Prof. Paul G. Tucker, Cambridge, email: pgt23@cam.ac.uk (expertise with roughness in gas turbines - simulations)

Section 9: Testbed Resources

Question #1

The ALCF and OLCF have test bed resources for new technologies, details below. If you would like access to these resources to support the work in this proposal, please provide the information below. (1 Page Limit)

The OLCF Quantum Computing User Program is designed to enable research by providing a broad spectrum of user access to the best available quantum computing systems, evaluate technology by monitoring the breadth and performance of early quantum computing applications, and Engage the quantum computing community and support the growth of the quantum information science ecosystems. More information can be found here: https://www.olcf.ornl.gov/olcf-resources/compute-systems/quantum-computing-user-program/quantum-computing-user-support-documentation.

The ALCF AI Testbed provides access to next-generation of AI-accelerator machines to enable evaluation of both hardware and workflows. Current hardware available includes Cerebras C-2, Graphcore MK1, Groq, Habana Gaudi, and SambaNova Dataflow. New hardware is regularly acquired as it becomes available. Up to date information can be found here: https://www.alcf.anl.gov/alcf-ai-testbed.

Describe the experiments you would be interested in performing, resources required, and their relationship to the current proposal. Please note, these are smaller experimental resources and a large amount of resources are not available. Instead, these resources are to explore the possibilities for these technologies might innovate future work. This request does not contribute to the 15-page proposal limit.

2023_INCITE_Testbed_Access.pdf
The attachment is on the following page.

We are not requesting access to Testbed Resources, although we are interested in learning more about whether our problem (solving the Navier-Stokes equations) can be addressed by new Quantum Computing Technology.