In-Tunnel CFD Simulation of the HL-CRM in the LaRC 14 x 22 ft. Wind Tunnel –
Part I: Empty Tunnel Simulation Approaches and Verification Using TAU-DRSM

R. Rudnik, S. Melber-Wilkending
DLR, Institute of Aerodynamics and Flow Technology
Braunschweig, Germany

109th NIA CFD Seminar, Hampton VA, 14.03.19
Objectives and background

In-tunnel simulation approaches

Numerical method
  - Flow solver, turbulence modelling, boundary conditions
  - Grid generation

Method verification / validation for empty tunnel computations

Conceptual variations of numerical set-up (inlet boundary condition, diffusor cutback)

Grid resolution study

Conclusion and next steps
Objective and background

- **AIAA High Lift Prediction Workshop Series**
  - initiated 2010 by Boeing and NASA
  - Objective: Assess numerical prediction capability of current-generation CFD algorithms/codes commercial aircraft high-lift configurations
  - Workshops in 2010 (Chicago), 2013 (San Diego), 2017 (Denver)
  - Preparation for the HiLiftPW IV (2021) ongoing
  - Present investigation intended to prepare 1 of 8 focus problems “in-tunnel runs” (HL-Common Research Model in NASA 14 x 22 ft. wind tunnel)

- **NATO AVT-284 Workshop on Advanced Wind Tunnel Boundary Simulation**
  - held in spring 2018 (Torino, Italy)
  - Objectives: Evaluate high-fidelity CFD simulation of wind tunnel wall boundaries, installation effects (e.g. support hardware) and compare with established results
  - Develop recommendations for the use of high fidelity simulation of wall boundaries and model support hardware
In-Tunnel Simulation Approaches

- Approach I: simulate full airline and all components
- Approach II: reduce to test section
- Approach III: simulate “essential” components (part of settling chamber, nozzle, test section, part of diffusor) adopted for DLR in-tunnel simulation approach
In-Tunnel Simulation Approaches

- **Approach III: simulate essential components**
  - pro: moderate numerical effort
  - pro: adequate specification and simulation of incoming boundary layer possible
  - pro: tunnel control inflow / outflow simulated according to real tunnel operation
  - con: inflow specification requires adaptation due to cut-off of the airline

- **Sensing and control of operational conditions elaborated**
  - Establish a numerical ‘device’ that allows b.l. to develop downstream of the last turbulence screen
  - Upstream flow condition sensors
  - Set-up sensor driven control loop
to adjust flow conditions by manipulating back pressure at the outlet
NASA LaRC 14 x 22 ft. High Speed Leg

1. diffusor

test section

honeycomb and 4 turbulence screens not modelled

nozzle

settling chamber

Sta. 0.0 17.75 50 191
TAU flow solver and features in use for present studies

- DLR TAU code is an edge-based, finite volume, unstructured flow solver (DV 2018.1.0)

- Full NS Discretization

- Spatial Discretization - main eq.: Jameson central, $2^{nd}$ order;
  blend scalar (20%) – matrix (80%) dissipation
  - turbulence eq.: AUSMDV upwind, $2^{nd}$ order

- Temp. Integration: - LU-SGS Backward Euler, 2V MG cycle

- Turbulence modeled by differential SSG/LRR RSM developed at DLR (Eisfeld, Braun)
  - pressure-strain correlation is modeled by a blend of correlations derived by
    Speziale, Sakar, Gatski and Launder, Reece, Rodi (for near wall treatment)
  - blending accomplished by F-function according to Menter’s BSL model
  - diffusion term modeled via generalized gradient diffusion model (Daly and Harlow)
  - $\omega$- based length scale equation blended acc. to Menter’s BSL formulation $(\omega, g)$
  - present computations based on new formulation $\tilde{\omega} = ln(\omega)$
    coupling to Reynolds stress equations using $\omega = e^{\tilde{\omega}}$
Sensor / b.c. setup to control adequate adjustment of flow conditions:

- Inlet
  - Constant pressure across inlet plane assumed - not fulfilled for x-location at turbulence screen #3 due to upstream effects of nozzle pressure field
  - Inlet plane artificially extended to justify assumption
  - p\(_t\) and T\(_t\) prescribed according to w/t specification of reference flow conditions
  - Adequate initializing wall boundary layer by introducing a numerical diverter controlled by outflow pressure (below diverter). Static pressure at b.c. adjusted to pressure at tunnel axis (relaxation introduced)
Sensor/b.c. setup to control adequate adjustment of flow conditions:

- Static ("back") pressure used to adjust flow conditions according to specification and compensate losses.

- Control loop scenario to adjust flow conditions based on nozzle pressure sensors including PT1 damper (controlling parameters and relaxation carefully adapted)

- Upstream pressure sensors (single probe or nozzle gradient)

- Numerical boundary condition to adjust flow conditions by static pressure at the outlet
Commonly used SOLAR features

- Developed by ARA, QinetiQ, Airbus, BAE Systems to provide hybrid unstructured grids
- Surface discretization: mixed element, quad-dominant mesh
- Volume discretization: hex-dominant mesh near aerodynamic surfaces
- $y^+$-manual adaptation, const. first cell height
- Special CAD-based treatment of grid refinement sources for improved surface discretization
Objective Verify DLR TAU computational setup for in-tunnel CFD computations against published results of 2015 ASM, Vigyan, NASA Study of 14 x 22 ft. high speed leg

Reference: AIAA 2015-2022

Numerical set-up used for AIAA 2015-2022:
- Flow solver: NASA TetrUSS (FUN3D, USM3D)
- SA-turbulence model
- Grids: VGRID, hybrid unstructured, 5.6 x 10^6 grid pts., y* = 0.5
- Boundary conditions Inlet: subsonic inflow / jet b.c. (USM3D)
  Outlet: backpressure b.c., p_s iterated to achieve target velocity

DLR computations:
- Flow solver: TAU, set-up as described before
- Grids: SOLAR hybrid unstructured, 28.0 x 10^6 grid pts. (“fine”), y* < 0.2
- Boundary conditions Inlet: subsonic inflow / jet b.c.
  Outlet: backpressure b.c., p_s iterated via controller

Flow cons: M = 0.2678  q = 101.5 lb/ft²  Re = 1.7108 x 10^6 per ft.
USM3D (SA) - FUN3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Mach contours, horizontal plane at \( z = 0.0 \)
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Mach contours, vertical plane at y = 0.0

Verification / Validation TAU ET vs. ASM, Vigyan, NASA Study (AIAA 2015-2022)
USM3D (SA) - FUN3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Centerline Mach profiles
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Longitudinal pressure distribution
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Wall-to-wall velocity profiles at \( x = 17.57 \) ft.
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Wall pressure distributions: test section ceiling
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- **Wall pressure distributions**: test section inner wall

![Graph showing wall pressure distributions for different rows in the test section.](image)
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Wall pressure distributions: test section outer wall
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Contours of upwash angle along test section at Sta 0.0
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Contours of upwash angle along test section at Sta 17.75
USM3D (SA) - TAU (DRSM) using fine grid / NASA intake / full diffusor

- Contours of upwash angle along test section at Sta 50
Objective: analyze the impact of variations in the geometrical and numerical set-up on the numerical results for the empty-tunnel simulation

Numerical set-up
- Flow solver: TAU, set-up as described before
- Grids: SOLAR hybrid unstructured, 28.0 x 10^6 grid pts. (“fine”), \( y^* = < 0.2 \)
- Boundary conditions Inlet: subsonic inflow
  Outlet: backpressure b.c., \( p_s \) iterated via controller

Variations considered
- DLR inlet: extension to account for homogeneous flow assumption
- Cut-back of the 1. diffusor from 192 ft. to 72 ft.

Flow cons: \( M = 0.2678 \) \( q = 101.5 \text{ lb/ft}^2 \) \( \text{Re} = 1.7108 \times 10^6 \text{ per ft.} \)
Conceptual Variations of Numerical Set-up - Geometry / B.C. Variants

NASA intake / full diffusor

Detail DLR intake

DLR intake / full diffusor

DLR intake / short diffusor
TAU (DRSM) using fine grid for variations of intake and diffusor

- Mach contours, vertical plane at $y = 0.0$
- Contours of upwash angle along test section at Sta 0.0

NASA intake / full diffusor

DLR intake / full diffusor

DLR intake / short diffusor
Contours of upwash angle along test section at Sta 17.75

- NASA intake / full diffusor
- DLR intake / full diffusor
- DLR intake / short diffusor
Conceptual Variations of Numerical Set-up - Geometry / B.C. Variants

- **Centerline Mach profiles**

  ![Mach profile graph]

- **Longitudinal pressure distribution**

  ![Pressure distribution graph]
• **Wall pressure distributions in test section outer wall**

![Graph showing wall pressure distributions in test section outer wall](image)
Objective: analyze the impact of grid resolution on the test section characteristics and diffusor flow

Numerical set-up: DLR Intake / full diffusor

4-level grid resolution study

<table>
<thead>
<tr>
<th>Grid</th>
<th>coarse</th>
<th>medium</th>
<th>fine</th>
<th>x-fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid Points</td>
<td>3.94 x 10^6</td>
<td>11.03 x 10^6</td>
<td>32.09 x 10^6</td>
<td>92.95 x 10^6</td>
</tr>
<tr>
<td>Hexahedra</td>
<td>3.60 x 10^6</td>
<td>10.14 x 10^6</td>
<td>29.43 x 10^6</td>
<td>84.96 x 10^6</td>
</tr>
<tr>
<td>Surface Elements</td>
<td>0.16 x 10^6</td>
<td>0.32 x 10^6</td>
<td>0.67 x 10^6</td>
<td>1.39 x 10^6</td>
</tr>
<tr>
<td>Total Elements</td>
<td>4.95 x 10^6</td>
<td>14.2 x 10^6</td>
<td>42.77 x 10^6</td>
<td>127.72 x 10^6</td>
</tr>
<tr>
<td>No. struct. Layers</td>
<td>29 + 9</td>
<td>40 + 9</td>
<td>57 + 9</td>
<td>81 + 9</td>
</tr>
<tr>
<td>1st wall spacing, y_n</td>
<td>2.080e-6 m</td>
<td>1.442e-6 m</td>
<td>1.000e-6 m</td>
<td>0.693e-6 m</td>
</tr>
<tr>
<td>y_n⁺</td>
<td>&lt; 0.42</td>
<td>&lt; 0.29</td>
<td>&lt; 0.2</td>
<td>&lt; 0.14</td>
</tr>
</tbody>
</table>
Grid Resolution Study - Longitudinal and Cross Section

Test section  Sta 17.75

- coarse
- medium
- fine
- x-fine
• **Mach contours, vertical plane at \( y = 0.0 \)**
Grid Resolution Study - Centerline Distributions

- **Centerline Mach profiles**

![Mach profile graph](image)

- **Longitudinal pressure distribution**

![Pressure distribution graph](image)
• Wall pressure distributions in test section outer wall
- **Contours of crossflow velocity at Sta 17.75**

Grid Resolution Study - Flow Field Results in Test Section

![Flow field results](image-url)
Grid Resolution Study - Flow Field Results at Diffusor Exit

- Contours of crossflow velocity at Sta 191

[Images of flow field results for coarse, medium, fine, and x-fine grid resolutions]
Grid Resolution Study - Flow Field Results in the Diffusor

coarse

medium

fine

x-fine
A numerical study of the empty high speed leg of the NASA 14 x 22 ft. low speed wind tunnel has been presented addressing different aspects of in-tunnel simulation.

The DLR TAU/SOLAR package has been used with a differential SSG-/LLR ln(ω) model.

An indirect validation and verification with numerical and experimental results from a previous study reveals close agreement to the existing CFD results and fair agreement to experimental evidence with discrepancies supposed to be caused by simplifications of geometrical wind tunnel details.

Vortex formation in the test section and diffuser is observed, which underlines the capability of the DRSM to resolve such type of phenomena.

Conceptual variations of numerical set-up w.r.t inlet boundary simulations and a diffusor cutback doesn’t affect the main characteristics of the flow in the test section.

- for the inlet modification some effects were observed in the uniformity of the test section inflow
- for the diffusor cut-back very minor differences were found due to the diffusor cut-back (empty section !)
Conclusion and next steps

- Grid resolution study featuring an order of magnitude variation in total grid point number and a widely identical grid topology shows basically no visible effect on the main characteristics of the flow in the (empty!) test section
  - for the vortex formation, effects of reduced numerical dissipation are found leading to more distinct vortices of reduced extent for higher resolved grids

Next steps:

- Integration of the HL-CRM WB CAD model into the tunnel set-up
- Grid generation based on an Chimera approach for the model vicinity with high a resolution near wall area
- Computation of the HL-CRM in-tunnel an free air using the DRSM
- Evaluation of results and assessment of installation impact on the aerodynamic characteristics
Thanks for your attention!
The authors like to acknowledge the support of NIA and NASA for conducting this research
Turbulence Model Study – SA Variants on fine grid level

SA

SA-QCR

SARC

SARC-QCR
Turbulence Model Study - SARC-QCR vs. SSG/LRR $\ln(\omega)$ on fine grid level