QNET-CFD WIKI
KNOWLEDGE BASE

UNDERLYING FLOW REGIME DOCUMENT TEMPLATE

UNDERLYING FLOW REGIME (UFR) DOCUMENT

ABSTRACT

Provide a summary of the UFR test-case submission.

1. INTRODUCTION

Give a brief overview of the UFR in question. Describe the main characteristics of the type of flow. In particular, what are the underlying flow physics which characterise this UFR and must be captured by the CFD methods?

2. REVIEW OF UFR STUDIES AND CHOICE OF TEST CASE

Provide a brief review of past studies of this UFR which have included test-case comparisons of CFD results involving models with experimental measurements or DNS results as target. Identify your chosen study (or studies) on which the document will focus. State the test-case underlying the study and briefly explain how well this represents the UFR? Give reasons for this choice (e.g a well-constructed test case, a recognised international comparison exercise, accurate measurements, reliable and well-documented DNS results, good quality control, a rich variety of turbulence or physical models assessed etc.) . If possible, the study should be taken from established data bases. Indicate whether or not the experiments have been designed for the purpose of CFD validation (desirable but not mandatory)?

3. BRIEF DESCRIPTION OF THE STUDY TEST CASE

This should:

- Convey the general set up of the test-case configuration (e.g. airflow over a bump on the floor of a wind tunnel)
- Describe the geometry, illustrated with a sketch
- Specify the flow parameters which define the flow regime (e.g. Reynolds number, Rayleigh number, angle of incidence etc.)
- Give the principal assessment quantities that were measured or obtained in a DNS by which the success or failure of CFD calculations are to be judged. These quantities should include global parameters but also the distributions of mean and turbulence quantities.

The description can be kept fairly short if a link can be made to a data base where details are given. For other cases a more detailed, fully self-contained description should be provided.

4. TEST CASE EXPERIMENTS/DNS

For experiments, provide a brief description of the test facility, together with the measurement techniques used. Indicate what quantities were measured and where.

Discuss the quality of the data and the accuracy of the measurements. It is recognized that the depth and extent of this discussion is dependent upon the amount and quality of information provided in the source documents. However, it should seek to address:

- How close is the flow to the target/design flow (e.g. if the flow is supposed to be two-dimensional, how well is this condition satisfied)?
- Estimation of the accuracy of measured quantities arising from given measurement technique
- Checks on global conservation of physically conserved quantities, momentum, energy etc.
- Consistency in the measurements of different quantities.

Discuss how well conditions at boundaries of the flow such as inflow, outflow, walls, far fields, free surface are provided or could be reasonably estimated in order to facilitate CFD calculations.

If target values for comparison are DNS results, refer to the relevant DNS test case in the KB Wiki or to a publication where the DNS study is fully described.

5. CFD METHODS

Provide an overview of the methods used to analyze the test case. This should describe the codes employed together with the turbulence/physical models examined; the models need not be described in detail if good references are available but the treatment used at the walls should explained. Comment on how well the boundary conditions used replicate the conditions in the test rig, e.g. inflow conditions based on measured data at the rig measurement station or reconstructed based on well-defined estimates and assumptions.

Discuss the quality and accuracy of the CFD calculations. As before, it is recognized that the depth and extent of this discussion is dependent upon the amount and quality of information provided in the source documents. However the following points should be addressed:

- What numerical procedures were used (discretisation scheme and solver)?
- What grid resolution was used? Were grid sensitivity studies carried out?
- Did any of the analyses check or demonstrate numerical accuracy?
- Were sensitivity tests carried out to explore the effect of uncertainties in boundary conditions?
- If separate calculations of the assessment parameters using the same physical model have been performed and reported, do they agree with one another?

6. COMPARISON OF CFD CALCULATIONS WITH EXPERIMENTS

Discuss how well the CFD calculations of the assessment quantities compare with experiment and with one another. Present some key comparisons in the form of tables or graphical plots and, where possible, provide hyperlinks to the appropriate results database. Results with different turbulence models covering as wide a range as possible should be included in the discussion. However, if too many different calculation results are available (e.g. from workshops) do not present all the comparisons here. A selection should be made showing results only for the most typical and practically important models. Comprehensive comparisons can be made available via a link to the associated databases. Finally, draw conclusions on the ability of the models used to simulate the test case flow.

7. BEST PRACTICE ADVICE FOR THE UFR

This section should be structured around the following subsections:

7.1 Key Physics
Summarise the key flow physics which characterise the UFR and which must be captured for accurate prediction of the assessment parameters

7.2 Numerical modelling issues
- Discretisation method
- Grids and grid resolution
- Boundary conditions and computational domain

7.3 Physical modelling
- Turbulence modelling
- Transition modelling
- Near-wall modelling
- Other modelling

7.4 Application uncertainties
Summarise any aspects of the UFR model set-up which are subject to uncertainty and to which the assessment parameters are particularly sensitive (e.g location and nature of transition to turbulence; specification of turbulence quantities at inlet; flow leakage through gaps etc.)

7.5 Recommendations for Future Work

Wherever possible, the advice should be in the form of an instruction rather than a conclusion. A conclusion could precede the 'instruction' in order to provide context. Thus, for example; 'The aerodynamic coefficients can be accurately predicted with algebraic turbulence models. However, these fail to predict the detailed dynamics of the wake boundary layer interaction. Such detail can, however, be predicted with reasonable accuracy using Spalart and Allmaras'is a conclusion. The BPA advice flowing from this conclusion is;

- *' Use algebraic turbulence models if the requirement is to predict accurately just the aerodynamic coefficents'*
- *'Use the Spalart Allmaras turbulence model if the requirement is to predict the detailed dynamics of the wake-boundary layer interaction as well as the aerodynamic coefficients.'*

It is generally easier to draw conclusions than to convert these into clear statements of advice. Thus it may be helpful to first set down your conclusions at the end of Section 6. 'Comparison of CFD Calculations with Experiments' and then work on these to develop the BPA in section 7..

Be extremely careful to ensure that your BPA is strongly supported by the evidence examined in Section 6. Do not offer advice based upon your own experience or prejudices or upon published/unpublished evidence which is not fully examined in the UFR document (e.g. you may have read a recent paper which concludes Spalart and Allmaras is the best for this test case. You cannot base BPA on this if you have not discussed the calculations here).

At the end of the BPA please add a sub-section 'Recommendations for Future Work'. This should propose further studies which will improve the quality or scope of the BPA and perhaps bring it up to date. For example, perhaps further calculations of the test-case should be performed employing more recent, highly promising models of turbulence (e.g Spalart and Allmaras, Durbin's v2f, etc.). Or perhaps new experiments should be undertaken for which the values of key parameters (e.g.

pressure gradient or streamline curvature) are much closer to those encountered in real application challenges.

8. REFERENCES

List references describing in detail the relevant measurements, turbulence models, numerical methods, CFD results etc.