3 EXPERIMENTAL AND COMPUTATIONAL PROCEDURES

3.1 Testing procedure

All testing of the valley and hill geometries was completed using the Seitz shock tube at the Mechanical Engineering Laboratory (University of the Witwatersrand). The following steps outline the procedure followed for each test undertaken throughout the duration of the research:

1. Pressurise air receiver to 10 bar, allowing sufficient time to reach operating conditions before testing commences (Approximately 30 minutes).
2. Check the supply pressure from the air receiver is sufficient for required Mach number (above 600 kPa).
3. Switch on all the instrumentation.
4. Check that the optics system is set-up correctly.
5. Place new film in the camera if needed.
6. Place test piece in test section and secure using three M6 bolts.
7. Remove any diaphragm fragments in the compression chamber and expansion chamber remaining from previous tests. This is achieved by means of ‘blow-down’ using a high-pressure air hose.
8. Vacuum-clean the inside of the expansion chamber to remove any diaphragm fragments that were not blown out by the high-pressure air. This is achieved by inserting an ordinary vacuum cleaner hose through the test section of the shock tube down the expansion chamber. (This step need only be done when felt necessary).
9. Place plastic diaphragms between the compression chamber and intermediate chamber and between the intermediate chamber and the expansion chamber. The correct diaphragm thickness is calculated by the personal computer. By entering the desired Mach number the personal computer calculates what diaphragm thickness is needed for the two positions in order to achieve the desired Mach number.
10. Close test section door and bolt securely shut. Make sure all secondary bolts are flush against the door otherwise tighten them until so.

11. Wind film to the next frame.
12. Enter required value of shock wave Mach number into the personal computer.
13. Enter required time delay into the personal computer.
14. Check all valves used for the shock tube are in the correct position.
15. Follow the instructions given by the personal computer to initiate test.
16. Record ambient conditions given by the personal computer.
17. Switch off all lights until test is completed.
18. Once the test has completed record the actual Mach number and test date and time.

3.2 Developing procedure

Once a sufficient amount of photographs have been taken on a spool, the film is developed in the dark room located in the Mechanical Engineering Laboratory (University of the Witwatersrand). The followings steps outline the procedure for developing of black and white photographs.

1. Ready all equipment needed for development. This includes rinsing all containers and measurement apparatus and placing all relevant components at arms length for easy accesses while the lights are switched off.
2. Prepare a basin of water, preferable at a temperature between 22 and 24 °C, for mixing of development chemicals.
3. Switch off lights and remove film from its casing.
4. Cut off unnecessary parts of the film for easier loading onto the spool.
5. Load film onto the spool, place spindle through the centre of the spool and then place in developing canister. Place primary lid securely.
6. Switch lights back on.
7. Mix 500 ml of the pre-prepared warm water with 20 ml of the developer (AGFA Rodinal) and pour into developing canister.
8. Secure the developing canister's secondary lid and knock against a flat surface to remove any air bubbles.
9. Agitate the developing mixture using the agitator shaft for one minute and then for 20 seconds every minute for the required developing time. (The developing time is based on

the temperature of the prepared warm water and can be determined from the graph shown in appendix D. For a temperature of 24 °C the developing time is four minutes.)

10. Once the developing time has elapsed rinse the developing canister, making sure that the primary lid is secure at all times.
11. Rinse measuring equipment again before measuring the required fixer mixture.
12. Mix 420 ml of the pre-prepared warm water with 60 ml of fixer and pour into developing canister.
13. Agitate the fixer mixture for one minute and then allow to stand for four minutes.
14. Once the fixing time has elapsed, rinse developing canister.
15. Remove the film from the spool.
16. Add two drops of wetting agent to the basin of prepared warm water and place film in the soapy water. Rub film down carefully to remove the sticky layer present on the film surface. Do so cautiously as not to scratch the surfaces.
17. Hang film up to dry.
18. Rinse the basin, measuring apparatus and all components of the developing canister. Then place to dry.

3.3 Experimental precautions

3.3.1 Shock tube operating precautions

1. Always make sure there is sufficient pressure in the supply tank.
2. Ensure that the test section door is open during the blown down procedure.
3. Ensure that the test section windows are free of finger prints and obstacles.
4. Always check that the optic system is set up correctly.
5. Ensure that the test section door is always securely shut before running a test.
6. Wear protective ear plugs during the blow down and shock generation procedures.

3.3.2 Film developing precautions

1. Ensure that the extraction fan is switched on to allow for sufficient ventilation.
2. Prepare all developing equipment before switching of the lights to remove film from its casing.

3. Remove or cover all light sources such as watch faces as not to over expose the film.
4. Take care as not to splash chemicals on clothes or skin as chemicals may leave stains and can be toxic.
5. Never unscrew the developing canister's primary lid until fixer phase is completed.

3.4 Computational Fluid Dynamics (CFD) modelling of the flow fields

The computational results were accomplished through the use of a Computational Fluid Dynamics (CFD) software programme, FLUENT. FLUENT is a program used for modeling fluid flow and heat transfer in complex geometries. The program provided complete mesh flexibility, solving flow problems with unstructured meshes that were generated about the complex geometries with relative ease. Two mesh types were used to obtain well resolved solutions, namely 3D hexahedral and mixed (hybrid) meshes. GAMBIT was used as the preprocessor for the geometry modeling and mesh generation, however the grid was refined or coarsened based on the flow solution in FLUENT.

The three-dimensional mesh for each valley and hill geometry was generated using GAMBIT and the mesh file imported into FLUENT where a three-dimensional inviscid, unsteady, coupled, explicit solver was used to model the fluid flow. The inviscid flow analysis, which solves the Euler equations, was used to provide a good initial solution for the problem due to its relatively complicated flow physics and complicated flow geometry. A time-dependent solution was required, therefore an unsteady solver using global time stepping was enabled. A coupled solver was selected to solve the governing equations of continuity, momentum, and energy and species transport simultaneously as a set, or vector, of equations. Because the governing equations are non-linear (and coupled), several iterations of the solution loop are performed before a converged solution is obtained. The chosen manner in which the governing equations were linearised was in explicit form with respect to the dependent variable of interest. This formulation was used to capture the transient behaviour of the incident shock wave. Due to the fact that the flow in most of the domain is not rotating an absolute velocity formulation was selected.

To set the fine-grid time step factor a Courant number of 0.85 was used to control the multi-stage time-stepping scheme in FLUENT. By entering a new refine/coarsen threshold the fully automated dynamic gradient adaption was initialised. The mesh adaption allowed the setting of limits on the minimum cell size, the minimum and maximum number of cells, and the cell types to be adapted. In addition, volume weighting of the gradient adaption function was also set. For a flexible mesh adaption a hanging node adaption was chosen. Meshes produced through this adaption are characterized by nodes on edges and faces that are not vertices of all the cells sharing those edges or faces. This mesh adaption also provided the ability to operate on grids with a variety of cell shapes, including hybrid meshes.

The complete flow solution was obtained in three-dimensional form. Two-dimensional images of the resulting flow were generated by taking slices of the flow field in the x-z and y-z planes. Three-dimensional images were generated to analyse and validate the existence of the complex three-dimensional wave structures.

3.5 Development of the three-dimensional model

The two-dimensional images obtained from CFD modelling were imported into a NURBS (non-uniform rational B-splines) modelling package (three-dimensional surface modelling), RHINOCEROS. The two-dimensional images, as well as a comprehensive understanding of the three-dimensional CFD images, were then used to generate the three-dimensional models. The procedure is outlined below.

1. Firstly, it was important to choose a specified rectangular area onto which the two-dimensional CFD images will be placed. The size of the image could be the actual size of the flow field, if known, or an arbitrary size suitable for imagery. If an arbitrary size was chosen then a known length of the image must be measured and used to scale other aspects of the flow.

2. A series of planes are created in the desired view-port, each separated by the distance between the images obtained from the two-dimensional CFD slices, and scaled using the scale value obtained in step 1. The shock features from the CFD images were then transferred onto these planes.

3. The two-dimensional images were imported into the modelling package using the “PlaceBackgroundBitmap” command and placed in the desired view plane.

4. The various features visible in the image such as the incident shock, reflected shock, Mach stem, etc, were traced using the various line commands onto the plane on which the image was placed. Different colours were assigned to the various surfaces by using the layer command box. This allowed for easier interpretation and creation of the surfaces.

5. The process was repeated until all of the desired CFD images had been utilised.

6. The ends of the transferred curves must be lined up to obtain smooth and continuous surfaces. This was done by zooming in as close as possible to the ends of the curves and dragging the curves either close to each other until all points matched up or until all end points were placed in a specific location.

7. The transferred curves were then used to generate the desired wave surfaces using the sweeping or lofting method.

8. The created surfaces were finally rendered, presenting a clear three-dimensional model of the shock waves surfaces.