A Thesis by

Mohammad Ali Badr

Bachelor of Science, Azad University, 2007

Submitted to the Department of Aerospace Engineering
and the faculty of the Graduate School of
Wichita State University
in partial fulfillment of
the requirements for the degree of
Master of Science

May 2011
SHOCK WAVE TURBULENT BOUNDARY LAYER INTERACTION
OVER A PROTRUSION

The following faculty members have examined the final copy of this thesis for form and content, and recommend that it be accepted in partial fulfillment of the requirement for the degree of Master of Science with a major in Aerospace Engineering.

______________________________
Klaus A. Hoffmann, Committee Chair

______________________________
Hamid Lankarani, Committee Member

______________________________
Roy Myose, Committee Member
DEDICATION

To my parents and sisters; whom I love the most
There is no wealth like knowledge,
no poverty like ignorance.
ACKNOWLEDGEMENTS

I would like to thank my adviser, Dr. Klaus Hoffmann, for his many years of thoughtful and patient guidance and support. Thanks are also due to members of my committee, Hamid Lankarani and Roy Myose, for their helpful comments and suggestions during all stages of this project. I also want to thank my friend and colleague, Armin Ghoddoussi, for all of his help, opinions, and collaborations.
ABSTRACT

This research attempts to investigate an important and common phenomenon in aerodynamics called shock interaction in a turbulent flow’s boundary layer. Due to advancements in current computational units, more complex geometries could be simulated with providing more accurate results. The tools used in this investigation are computational turbulent model of hybrid RANS/LES, called detached eddy simulation (DES). DES and its variant delayed detached eddy simulation (DDES) were the two computational schemes used for numerical simulation. Two protrusions were focused on in this work: a symmetrical bump and a proposed aircraft UHF antenna. Computation where performed with commercial software Cobalt and FLUENT in the High Performance Computing Center (HiPeCC) in Wichita State University. Computational simulation is costly in terms of energy consumption and time usage. Even so with the advanced computational units of HiPeCC, using in average of 18 processors, total simulation for this research took over 2 months of simulation.
PREFACE

As part of the master’s degree requirement at Wichita State University, a graduate student must conduct research on a subject agreed on by a faculty member and present the results of that research in front of a thesis committee. The research in this thesis began with the author’s education in computational fluid dynamics (CFD) and the fundamentals of aerodynamics and fluid dynamics. A test case was selected to practice using this knowledge, and finally, an actual case was solved and published. Results from the simulated cases were presented at the 49th AIAA Aerospace Sciences Meeting (2011) in Orlando, Florida, under the title “Detached Eddy Simulation of Shock Turbulent Boundary Layer Interaction over a Protrusion,” by Mohammad A. Badr and Klaus A. Hoffmann.
# TABLE OF CONTENTS

Chapter | Page
--- | ---
1. INTRODUCTION | 1
1.1 Shock Wave Boundary Layer Interaction | 1
1.1.1 Flow Condition | 2
1.1.2 Flow Equations | 3
1.1.3 Flow Solvers and Turbulent Models | 4
1.2 Past Work | 10
1.2.1 Historical Research | 11
1.2.2 Numerical Research | 12
1.2.3 Experimental Research | 14
1.3 Computational Software | 15
2. MESH GENERATION | 16
2.1 Volume Mesh | 16
2.1.1 Gambit | 16
2.1.2 Gridgen | 18
2.2 General DES Mesh Condition | 19
2.3 Object and Computational Domain | 21
2.3.1 Bump | 21
2.3.2 Antenna | 24
3. SOLUTION PROCEDURE | 28
3.1 Bump | 28
3.2 Antenna | 30
4. RESULTS | 32
4.1 Bump | 32
4.1.1 2D Solution | 32
4.1.2 3D Bump Results Comparison | 36
4.1.3 3D Bump Expanded Results | 42
4.2 Antenna | 50
4.2.1 DES/DDES Antenna | 51
4.2.2 DDES Antenna with Side Slip | 64
5. CONCLUSIONS AND FUTURE WORK | 80
REFERENCES | 84
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Comparison of Approaches to Navier-Stokes Equation Solutions</td>
<td>4</td>
</tr>
<tr>
<td>2.</td>
<td>Numerical Investigations of SWBLI and SWTBLI</td>
<td>12</td>
</tr>
<tr>
<td>3.</td>
<td>Experimental Investigations of SWBLI and SWTBLI</td>
<td>14</td>
</tr>
<tr>
<td>4.</td>
<td>Bump X-Z Location in Millimeters</td>
<td>22</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Shock wave turbulent boundary interaction scheme</td>
<td>12</td>
</tr>
<tr>
<td>2.</td>
<td>Hexahedron volume mesh patterns: (a) 8 nodes, (b) 20 nodes, and (c) 27 nodes</td>
<td>17</td>
</tr>
<tr>
<td>3.</td>
<td>Wedge volume mesh patterns: (a) 6 nodes, (b) 15 nodes, and (c) 18 nodes</td>
<td>17</td>
</tr>
<tr>
<td>4.</td>
<td>Tetrahedron volume mesh patterns: (a) 4 nodes, (b) 10 nodes</td>
<td>18</td>
</tr>
<tr>
<td>5.</td>
<td>Pyramid volume mesh patterns: (a) 5 nodes, (b) 13 nodes, and (c) 14 nodes</td>
<td>18</td>
</tr>
<tr>
<td>6.</td>
<td>Bump view from plane slice</td>
<td>22</td>
</tr>
<tr>
<td>7.</td>
<td>Full bump model for wind tunnel</td>
<td>22</td>
</tr>
<tr>
<td>8.</td>
<td>Computational domain of bump protrusion</td>
<td>23</td>
</tr>
<tr>
<td>9.</td>
<td>DM-C34 series airborne UHF SATCOM antenna</td>
<td>25</td>
</tr>
<tr>
<td>10.</td>
<td>Antenna top view</td>
<td>25</td>
</tr>
<tr>
<td>11.</td>
<td>Antenna side view</td>
<td>26</td>
</tr>
<tr>
<td>12.</td>
<td>Antenna computational domains</td>
<td>26</td>
</tr>
<tr>
<td>13.</td>
<td>Mesh around antenna with a plane of y = 0 slice</td>
<td>27</td>
</tr>
<tr>
<td>14.</td>
<td>Surface mesh near antenna protrusion</td>
<td>27</td>
</tr>
<tr>
<td>15.</td>
<td>Schematic of boundary layer</td>
<td>29</td>
</tr>
<tr>
<td>16.</td>
<td>Flow on lower surface: (a) RANS-SA, (b) numerical and experimental data from</td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>Bron</td>
<td></td>
</tr>
<tr>
<td>17.</td>
<td>Bump $y^+$</td>
<td>34</td>
</tr>
<tr>
<td>18.</td>
<td>Static pressure contours</td>
<td>34</td>
</tr>
<tr>
<td>19.</td>
<td>Mach contours</td>
<td>35</td>
</tr>
<tr>
<td>20.</td>
<td>Turbulent viscosity contours</td>
<td>35</td>
</tr>
<tr>
<td>21.</td>
<td>Velocity angle contours</td>
<td>36</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>--------</td>
</tr>
<tr>
<td>22.</td>
<td>Comparison of static pressure on lower surface at 10 mm from side wall: (a) DES-SA and experimental data, (b) ( k-\omega ) and experimental data from Bron</td>
<td>37</td>
</tr>
<tr>
<td>23.</td>
<td>Comparison of static pressure on lower surface at 25 mm from side wall: (a) DES-SA and experimental data, (b) ( k-\omega ) and experimental data from Bron</td>
<td>38</td>
</tr>
<tr>
<td>24.</td>
<td>Comparison of static pressure on lower surface at 50 mm from side wall: (a) DES-SA and experimental data, (b) ( k-\omega ) and experimental data from Bron</td>
<td>39</td>
</tr>
<tr>
<td>25.</td>
<td>Spanwise ( y^+ ) value distribution on the lower surface</td>
<td>40</td>
</tr>
<tr>
<td>26.</td>
<td>( y^+ ) value of lower surface compared with: (a) 1 cm from side wall, (b) 5 cm from side wall</td>
<td>41</td>
</tr>
<tr>
<td>27.</td>
<td>Streamlines near lower surface with static pressure contours</td>
<td>42</td>
</tr>
<tr>
<td>28.</td>
<td>Turbulent viscosity contours: (a) 1 cm, (b) 2.5 cm, (c) 5 cm from side wall</td>
<td>44</td>
</tr>
<tr>
<td>29.</td>
<td>Velocity angles (deg): (a) 1 cm, (b) 2.5 cm, (c) 5 cm from side wall</td>
<td>46</td>
</tr>
<tr>
<td>30.</td>
<td>Static pressure in the computational domain at: (a) 1 cm, (b) 2.5 cm, and (c) 5 cm from side wall</td>
<td>48</td>
</tr>
<tr>
<td>31.</td>
<td>Mach contours in the computational domain at: (a) 1 cm, (b) 2.5 cm, and (c) 5 cm from side wall</td>
<td>49</td>
</tr>
<tr>
<td>32.</td>
<td>Streamlines and turbulent viscosity</td>
<td>50</td>
</tr>
<tr>
<td>33.</td>
<td>Static pressure at ( Z = 10 ) cm, ( t = 0.15 ) s: (a) DDES, (b) DES</td>
<td>51</td>
</tr>
<tr>
<td>34.</td>
<td>Static pressure at ( Z = 20 ) cm, ( t = 0.15 ) s: (a) DDES, (b) DES</td>
<td>51</td>
</tr>
<tr>
<td>35.</td>
<td>Static pressure at ( Z = 26 ) cm, ( t = 0.15 ) s: (a) DDES, (b) DES</td>
<td>52</td>
</tr>
<tr>
<td>36.</td>
<td>Static pressure at ( Z = 28 ) cm, ( t = 0.15 ) s: (a) DDES, (b) DES</td>
<td>52</td>
</tr>
<tr>
<td>37.</td>
<td>Static pressure at ( Z = 30 ) cm, ( t = 0.15 ) s: (a) DDES, (b) DES</td>
<td>52</td>
</tr>
<tr>
<td>38.</td>
<td>Static pressure at ( Z = 10 ) cm, ( t = 0.3 ) s: (a) DDES, (b) DES</td>
<td>53</td>
</tr>
<tr>
<td>39.</td>
<td>Static pressure at ( Z = 20 ) cm, ( t = 0.3 ) s: (a) DDES, (b) DES</td>
<td>53</td>
</tr>
<tr>
<td>40.</td>
<td>Static pressure at ( Z = 26 ) cm, ( t = 0.3 ) s: (a) DDES, (b) DES</td>
<td>53</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>41.</td>
<td>Static pressure at $Z = 28$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>54.</td>
</tr>
<tr>
<td>42.</td>
<td>Static pressure at $Z = 30$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>54.</td>
</tr>
<tr>
<td>43.</td>
<td>Temperature at $Z = 10$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>54.</td>
</tr>
<tr>
<td>44.</td>
<td>Temperature at $Z = 20$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>55.</td>
</tr>
<tr>
<td>45.</td>
<td>Temperature at $Z = 26$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>55.</td>
</tr>
<tr>
<td>46.</td>
<td>Temperature at $Z = 28$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>55.</td>
</tr>
<tr>
<td>47.</td>
<td>Temperature at $Z = 30$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>56.</td>
</tr>
<tr>
<td>48.</td>
<td>Temperature at $Z = 10$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>56.</td>
</tr>
<tr>
<td>49.</td>
<td>Temperature at $Z = 20$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>56.</td>
</tr>
<tr>
<td>50.</td>
<td>Temperature at $Z = 26$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>57.</td>
</tr>
<tr>
<td>51.</td>
<td>Temperature at $Z = 28$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>57.</td>
</tr>
<tr>
<td>52.</td>
<td>Temperature at $Z = 30$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>57.</td>
</tr>
<tr>
<td>53.</td>
<td>Turbulent Viscosity at $Z = 10$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>58.</td>
</tr>
<tr>
<td>54.</td>
<td>Turbulent Viscosity at $Z = 20$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>58.</td>
</tr>
<tr>
<td>55.</td>
<td>Turbulent Viscosity at $Z = 26$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>58.</td>
</tr>
<tr>
<td>56.</td>
<td>Turbulent Viscosity at $Z = 28$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>59.</td>
</tr>
<tr>
<td>57.</td>
<td>Turbulent viscosity at $Z = 30$ cm, $t = 0.15$ s: (a) DDES, (b) DES</td>
<td>59.</td>
</tr>
<tr>
<td>58.</td>
<td>Turbulent viscosity at $Z = 10$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>59.</td>
</tr>
<tr>
<td>59.</td>
<td>Turbulent viscosity at $Z = 20$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>60.</td>
</tr>
<tr>
<td>60.</td>
<td>Turbulent viscosity at $Z = 26$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>60.</td>
</tr>
<tr>
<td>61.</td>
<td>Turbulent viscosity at $Z = 28$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>60.</td>
</tr>
<tr>
<td>62.</td>
<td>Turbulent viscosity at $Z = 30$ cm, $t = 0.3$ s: (a) DDES, (b) DES</td>
<td>61.</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES (continued)

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>63.</td>
<td>Streamlines in turbulent region with turbulent viscosity contours</td>
</tr>
<tr>
<td>64.</td>
<td>Turbulent Viscosity for $t = 0.4$ s and (a) $x = -10$ cm, (b) $x = -1$ cm, (c) $x = 10$ cm, (d) $x = 20$ cm, (e) $x = 30$ cm, (f) $x = 40$ cm, (g) $x = 50$ cm, (h) $x = 80$ cm, (i) $x = 90$ cm, (j) $x = 100$ cm</td>
</tr>
<tr>
<td>65.</td>
<td>Static pressure contours at $Z = 10$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>66.</td>
<td>Static pressure contours at $Z = 10$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>67.</td>
<td>Static pressure contours at $Z = 20$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>68.</td>
<td>Static pressure contours at $Z = 20$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>69.</td>
<td>Static pressure contours at $Z = 26$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>70.</td>
<td>Static pressure contours at $Z = 26$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>71.</td>
<td>Static pressure contours at $Z = 28$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>72.</td>
<td>Static pressure contours at $Z = 28$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>73.</td>
<td>Static pressure contours at $Z = 30$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>74.</td>
<td>Static pressure contours at $Z = 30$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>75.</td>
<td>Temperature contours at $Z = 10$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>76.</td>
<td>Temperature contours at $Z = 10$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>77.</td>
<td>Temperature contours at $Z = 20$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>78.</td>
<td>Temperature contours at $Z = 20$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>79.</td>
<td>Temperature contours at $Z = 26$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>80.</td>
<td>Temperature contours at $Z = 26$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>81.</td>
<td>Temperature contours at $Z = 28$ cm, $t = 0.15$ s</td>
</tr>
<tr>
<td>82.</td>
<td>Temperature contours at $Z = 28$ cm, $t = 0.3$ s</td>
</tr>
<tr>
<td>83.</td>
<td>Temperature contours at $Z = 30$ cm, $t = 0.15$ s</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES (continued)

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>84</td>
<td>Temperature contours at $Z = 30$ cm, $t = 0.3$ s</td>
<td>74</td>
</tr>
<tr>
<td>85</td>
<td>Turbulent viscosity contours at $Z = 10$ cm, $t = 0.15$ s</td>
<td>75</td>
</tr>
<tr>
<td>86</td>
<td>Turbulent viscosity contours at $Z = 10$ cm, $t = 0.3$ s</td>
<td>75</td>
</tr>
<tr>
<td>87</td>
<td>Turbulent viscosity contours at $Z = 20$ cm, $t = 0.15$ s</td>
<td>76</td>
</tr>
<tr>
<td>88</td>
<td>Turbulent viscosity contours at $Z = 20$ cm, $t = 0.3$ s</td>
<td>76</td>
</tr>
<tr>
<td>89</td>
<td>Turbulent viscosity contours at $Z = 26$ cm, $t = 0.15$ s</td>
<td>77</td>
</tr>
<tr>
<td>90</td>
<td>Turbulent viscosity contours at $Z = 26$ cm, $t = 0.3$ s</td>
<td>77</td>
</tr>
<tr>
<td>91</td>
<td>Turbulent viscosity contours at $Z = 28$ cm, $t = 0.15$ s</td>
<td>78</td>
</tr>
<tr>
<td>92</td>
<td>Turbulent viscosity contours at $Z = 28$ cm, $t = 0.3$ s</td>
<td>78</td>
</tr>
<tr>
<td>93</td>
<td>Turbulent viscosity contours at $Z = 30$ cm, $t = 0.15$ s</td>
<td>79</td>
</tr>
<tr>
<td>94</td>
<td>Turbulent viscosity contours at $Z = 30$ cm, $t = 0.3$ s</td>
<td>79</td>
</tr>
</tbody>
</table>
### LIST OF ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>Two-Dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>Three-Dimensional</td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Lewy</td>
</tr>
<tr>
<td>CUBRC</td>
<td>Calspan-University at Buffalo Research Center</td>
</tr>
<tr>
<td>D</td>
<td>Diameter</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>DES-SA</td>
<td>Detached Eddy Simulation with Spalart-Allmaras</td>
</tr>
<tr>
<td>DDES</td>
<td>Delayed Detached Eddy Simulation</td>
</tr>
<tr>
<td>DDES-SA</td>
<td>Delayed Detached Eddy Simulation with Spalart-Allmaras</td>
</tr>
<tr>
<td>DLR</td>
<td>Deutsches Zentrum für Luft- und Raumfahrt (German Aerospace Center)</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>DR</td>
<td>Departure Region</td>
</tr>
<tr>
<td>ER</td>
<td>Euler Region</td>
</tr>
<tr>
<td>FANS</td>
<td>Favre Average Navier-Stokes</td>
</tr>
<tr>
<td>FR</td>
<td>Focus Region</td>
</tr>
<tr>
<td>H</td>
<td>Height</td>
</tr>
<tr>
<td>HTT</td>
<td>High Temperature Tunnel</td>
</tr>
<tr>
<td>ITAM</td>
<td>Institute of Theoretical and Applied Mechanics</td>
</tr>
<tr>
<td>KTH</td>
<td>Kungliga Tekniska Högskolan (Royal Institute of Technology)</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LR</td>
<td>LES Region</td>
</tr>
<tr>
<td>NS</td>
<td>Navier-Stokes</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Average Navier-Stokes</td>
</tr>
<tr>
<td>RANS-SA</td>
<td>Reynolds Average Navier-Stokes with Spalart-Allmaras</td>
</tr>
<tr>
<td>RR</td>
<td>RANS Region</td>
</tr>
<tr>
<td>RWG</td>
<td>Rohrwindkanal Göttingen (Ludwieg Tube Göttingen)</td>
</tr>
<tr>
<td>SA</td>
<td>Spalart-Allmaras</td>
</tr>
<tr>
<td>SATCOM</td>
<td>Satellite Communication</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Implicit Method for Pressure-Linked Equations</td>
</tr>
<tr>
<td>SIMPLEC</td>
<td>Semi-Implicit Method for Pressure-Linked Equations, Consistent</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>SWBLI</td>
<td>Shock Wave Boundary Layer Interaction</td>
</tr>
<tr>
<td>SWTBLI</td>
<td>Shock Wave Turbulent Boundary Layer Interaction</td>
</tr>
<tr>
<td>TVD</td>
<td>Total Variation Diminishing</td>
</tr>
<tr>
<td>UB</td>
<td>University at Buffalo</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds Average Navier-Stokes</td>
</tr>
<tr>
<td>UHF</td>
<td>Ultra High Frequency</td>
</tr>
<tr>
<td>WENO</td>
<td>Weighted Essentially Non-Oscillatory</td>
</tr>
</tbody>
</table>
CHAPTER 1
INTRODUCTION

1.1 Shock Wave Boundary Layer Interaction

As the advances in aerospace engineering continue, it is more likely that investigation will take place on different sections and aspects of fluids versus objects and bodies. One such aspect is the interaction between a shock wave and the boundary layer (BL) of an object. This can be in two major formats: flow inside a body/channel (internal flow); or flow over an object, protrusion, or convection corner (external flow). In many cases, this is a critical point in aerodynamics or propulsion designing of such objects as engine inlets, external body elements, and materials, because the flow properties such as compressibility, turbulence, viscous-inviscid interaction phenomena, temperature, pressure, stress, etc., change sharply in the interaction regions [1].

The strategy to designing more suitable and efficient flying objects is either improving structural materials or improving aerodynamic designs. Today, with the advances in wind tunnels and computational methods, aerodynamic design is becoming more sophisticated and, in many cases, focuses in regions that were previously impossible. Research is being conducted in order to understand the interaction between a shock wave and the boundary layer, and to generate more efficient (less time- and energy-consuming) computational models.

Shock wave boundary layer interaction (SWBLI) occurs wherever there is a shock wave close to a body, i.e. at the boundary layer. Therefore, the velocity required begins at transonic speeds and increases up to hypersonic speeds. Therefore, most aeronautical or astronautical objects—from an ordinary passenger jet, to missiles and fighter jets, to space launchers and trans-atmospheric objects—must deal with this problem in all or some part of their flight paths.
1.1.1 Flow Conditions

The work in this thesis focused on flow that is primarily continuous, compressible, and viscous, meaning the following: (a) flow particles, or molecules, move around the body in such a way that there is never a gap between them and the body; (b) density changes as a certain outer pressure is forced to a constant amount of volume, or flow resistance increases as the particle moves faster within the flow; and (c) friction, thermal conduction, or diffusion is involved in the flow [2].

Fluids are also categorized as laminar, transitional, or turbulent. In the case of flow passing over a flat plate, layers of flow move in such a way that no significant amount of flow mass moves perpendicular to the rest of the flow set, or the flow path is distinguishable in a way that a non-body reaction does not notably change the flow pattern dramatically, then the flow is considered to be laminar. If the flow has an irregular fluctuation in its motion such that it creates eddies or mixture, then the state of flow is called turbulent [3]. In general if any flow characteristics have a notable variation regardless of an external force, the flow is turbulent. A flow that is between the laminar state and turbulent state is called transitional.

Most engineering problems involve turbulent flow. Some of the most significant characteristics of the turbulent flows are: (a) the general swirling of the flow creates crescent-shaped sections, or stream lines, called eddies; (b) the flow patterns become unsteady, (c) turbulence is mostly three dimensional; (d) the irregular variations of motion with respect to time or space are extremely large [4]. If the flow is laminar, then the shock interaction with the boundary layer is called the shock wave boundary layer interaction. If the flow is turbulent, then the shock wave interaction with the boundary layer is called the shock wave turbulent boundary layer interaction, (SWTBLI).
The final classification used in this thesis is one according to the Mach number. For a free stream Mach number less than 0.3, the flow is subsonic incompressible. If the Mach number is larger than 0.3 and less than 0.8, then it is called subsonic compressible. Flow with a Mach number larger than 0.8 and smaller than 1 is called transonic. Flow with a Mach number larger than 1 and smaller than 5 (or 8, depending on whether or not the chemical reactions are being considered) is called supersonic. And, for a high Mach number flow that is larger than 5 and the flow has a chemical reaction, the flow is called hypersonic. This thesis focused on transonic flow, but some supersonic flows were also examined.

1.1.2 Flow Equations

In a fluid system, like all dynamic systems, there are three basic laws: conservation of mass, conservation of energy, and conservation of momentum.

- Conservation of Mass: when applied to a point inside a continuous mass flux, this will resolve to the continuity equation:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0
\]  

(1)

- Conservation of Energy:

\[
\frac{\partial}{\partial t} \left[ \rho \left( e + \frac{V^2}{2} \right) \right] + \nabla \cdot \left[ \rho \left( e + \frac{V^2}{2} \right) \mathbf{V} \right] = \rho \dot{q} - \nabla \cdot (p \mathbf{V}) + \rho (\mathbf{f} \cdot \mathbf{V}) + \dot{Q}_{\text{VISCOUS}} + \dot{W}_{\text{VISCOUS}}
\]  

(2)

- Conservation of Momentum:

\[
\frac{\partial (\rho \mathbf{V})}{\partial t} + \nabla \cdot \left[ (\rho \mathbf{V} \times \mathbf{V}) + p \mathbf{I} \right] = \nabla \cdot \mathbf{t}
\]  

(3)

These equations were first discovered in the 19\textsuperscript{th} century by: Claude Louis Marie Henri Navier (in 1823) and Sir George Gabriel Stokes (in 1845) (reported by White [5]); thus, in their honor, these equations are called the “Navier-Stokes” (NS) equations.
1.1.3 Flow Solvers and Turbulent Models

The Navier-Stokes equations are partial differential equations, and for even the simplest two-dimensional case, it is very difficult (not to mention impossible) to achieve an exact answer. Like all engineering or mathematical problems, there are three solution types: analytical or direct, experimental simulation, and numerical calculation. Each method has its pros and cons. An illustrated comparison is presented in Table 1.

<table>
<thead>
<tr>
<th>Approach</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theoretical</td>
<td>Is clean, involves general information usually in formula form</td>
<td>Is restricted to simple geometry and physics, usually applies to linear problems</td>
</tr>
<tr>
<td>Experimental</td>
<td>Is capable of being most realistic</td>
<td>Requires equipment, has scaling problems, requires tunnel corrections, encounters measurement difficulties, has operating costs</td>
</tr>
<tr>
<td>Computational</td>
<td>Is not restricted to linearity, can be used to treat complicated physics, is possible to obtain time evolution of flow</td>
<td>Has truncation errors, encounters boundary condition problems, has associated computer costs</td>
</tr>
</tbody>
</table>

As mentioned earlier, fluid flow motion is very complex. More complexity occurs when the flow is in a turbulent state. Although NS equations are used to define fluid motion, it is essential to find a pattern that could manage flow properties in a turbulent regime.

Jimenez [7] provided a more in-depth discussion of the features of numerical and experimental simulations. The first mathematical approach was reported in 1877 by Boussinesq. In his model, the concept of eddy viscosity was introduced for the first time. In 1898, Reynolds used the time-averaged Navier-Stokes equation in his research on turbulence. But not much was known about the physics of viscous flow until 1904, when Prandtl discovered the boundary layer.
Later in 1925, Prandtl introduced the mixing length and, from that, a clear procedure to calculating eddy viscosity in its own terms. Every turbulent model in the twenty years following was based on this hypothesis. In 1930 von Karman theorized turbulent shear and in 1945, Prandtl introduced a turbulent model that related eddy viscosity with the turbulent flow’s kinematic energy, $k$. The idea here was that turbulence is related to where the flow had been, or the flow history. To calculate $k$ exactly, he introduced a partial differential equation. This solution needed to specify a turbulent length scale. By definition, this type of solution is called “incomplete.” A “complete” solution is one that does not need to have any advanced knowledge of any fluid property. The first complete solution was introduced by Kolmogorov in 1942, when in addition to the turbulent kinematic energy, $k$, he introduced another parameter, $\omega$, as

$$\omega = \frac{d\dot{E}}{d\nu dt} \quad (4)$$

where $d\dot{E}$ is the energy dissipation rate, $\nu$ is volume, and $t$ is time [8].

The approaches mentioned above are classified as “eddy viscosity” models, where the quantities are expressed either in terms of the mean velocity flow field or by the mean turbulent field. Another class of models is the “Reynolds stress” model, where closure is obtained by expressing the modeled quantities in terms of the mean turbulent field [9].

Turbulence is an unsteady state (varies in time) and has a highly catastrophic nature. The actual result can only be solved by finding the averages of the NS solutions. In this way, there are two types of averaging: time averaging and mass averaging. Time-averaging equations are also known as Reynolds average Navier-Stokes (RANS) equations, and mass-averaging equations are known as Favre average Navier-Stokes (FANS) equations. FANS equations are more likely to be used for Mach numbers higher than 5 [10].
Eddy viscosity and Reynolds stress models are two types of turbulent models used in RANS equations. If only one additional equation is required for calculating the turbulent energy, then it is called a one-equation turbulent model, like Prandtl’s incomplete solution. If two additional equations are required, like Kolmogorov’s $k-\omega$, then the model is considered a two-equation turbulent model. If no additional equations are required, like Boussinesq’s mixing length hypothesis, then the model is called a zero-equation model or analytical turbulent model.

One of the most common turbulence models is the one-equation turbulent model introduced by Spalart and Allmaras [11], which is now known as the Spalart-Allmaras (SA) turbulence model. The SA model is the main turbulence model used in this thesis.

Using Hoffmann and Chiang’s [10] format, the SA model can be driven as follows: Eddy viscosity, $\nu_t$, is calculated by

$$\nu_t = \bar{v} f_{v1} \quad (5)$$

where

$$f_{v1} = \frac{\chi^3}{\chi^3 + \kappa_1^3} \quad (6)$$

$$\chi = \frac{\bar{v}}{v} \quad (7)$$

$\bar{v}$ is calculated by

$$\frac{d\bar{v}}{dt} = \left(1 + \frac{c_{b2}}{\sigma}\right) \nabla \cdot [(v + \bar{v})\nabla \bar{v}] - \frac{c_{b2}}{\sigma} (v + \bar{v}) \nabla^2 \bar{v} + c_{b1}(1 - f_{t2}) \bar{v} \bar{S} - \left[ c_{w1} f_w - \frac{c_{b1}}{\kappa^2} f_{t2} \right] \left[ \frac{\bar{v}}{d} \right]^2 + f_{t1}(\Delta q)^2 \quad (8)$$

where $\kappa$ is the von Karman constant, and $d$ is distance to the wall. Other parameters can be calculated as

$$\bar{S} = S + \frac{v}{\kappa^2 d^2} f_{v2} \quad (9)$$

where $S$ is the magnitude of vorticity calculated by
\[ S = \left| \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right| \]  

(10)

and

\[ f_{u2} = \frac{\chi}{1 + \chi f_{u1}} \]  

(11)

Other equations are

\[ f_w(r) = g \left( \frac{1 + c_{w2}}{g^6 + c_{w3}^6} \right)^{\frac{1}{6}} \]  

(12)

\[ g = r + c_{w2}(r^6 - r) \]  

(13)

\[ r = \frac{\varphi}{\delta \kappa^2 d^2} \]  

(14)

\[ f_{t1} = c_{t1} g_t \exp \left[ -c_{t2} \left( \frac{w_t}{\Delta q} \right)^2 (d^2 + g_t^2 d_t^2) \right] \]  

(15)

\[ f_{t2} = c_{t3} \exp(-c_{t4} \chi^2) \]  

(16)

and the variables are as follows:

\[ d_t = \text{distance between point in the field and point of trip, located on the surface.} \]

\[ w_t = \text{wall vorticity at trip.} \]

\[ \Delta q = \text{difference between field point and trip point velocity.} \]

\[ g_t = \min \left[ 1.0, \frac{\Delta q}{w_t \Delta x} \right] \text{ and } \Delta x \text{ is grid spacing along the wall at the trip.} \]

Constants used in these equations are as follows:

\[ \sigma = \frac{2}{3}, \quad c_{b1} = 0.1355, \quad c_{b2} = 0.622, \quad \kappa = 0.41, \quad c_{w1} = \frac{c_{b1}}{\kappa^2} + \frac{1 + c_{b2}}{\sigma}, \]

\[ c_{w2} = 0.3, \quad c_{w3} = 2, \quad c_{v1} = 7.1, \quad c_{t1} = 1.0, \quad c_{t2} = 2.0, \quad c_{t3} = 1.1, \quad c_{t4} = 2.0. \]

It is safer to substitute \( c_{t3} \) and \( c_{t4} \) in the original case as follows: \( c_{t3} = 1.2, \quad c_{t4} = 0.5. \)

If the flow is compressible, then

\[ \mu_t = \rho \bar{v} f_{u1} \]  

(17)
where \( \rho \) is the local density, and the term \( \frac{d\bar{v}}{dt} \) is changed to \( \frac{\partial \bar{v}}{\partial t} + \frac{\partial (\bar{v}u_j)}{\partial x_j} \). Thus,

\[
\frac{\partial \bar{v}}{\partial t} + \frac{\partial (\bar{v}u_j)}{\partial x_j} = \left[ 1 + \frac{c_{b_2}}{\sigma} \right] \nabla \cdot [(\bar{v} + \bar{v}) \nabla \bar{v}] - \frac{c_{b_2}}{\sigma} (\bar{v} + \bar{v}) \nabla^2 \bar{v} + c_{b_1} (1 - f_{t_2}) \bar{v} \nabla^2 - \left[ c_{w_1} f_w - \frac{c_{b_3}}{k_2} f_{t_2} \right] \left[ \frac{\partial \bar{v}}{\partial t} \right]^2 + f_{t_1} (\Delta q)^2 \quad (18)
\]

The initial condition for this equation is

\[
\bar{v} = 0 \rightarrow \frac{\nu}{10}
\]

and the boundary conditions could be as follows:

Wall: \( \bar{v} = 0 \)

Inflow: \( \bar{v} = \bar{v}_\infty \)

Outflow: extrapolation would be used.

Beside RANS and FANS, other techniques are used to calculate and simulate turbulent flow: large eddy simulation (LES), detached eddy simulation (DES) and direct numerical simulation (DNS).

The approach for LES began in 1963, when Smagorinsky introduced his eddy viscosity. Lilly (his collaborator), was the first person to publish a LES of buoyant atmosphere using Smagorinsky’s eddy viscosity. Deardorff used LES in fluid dynamics for the first time as reported by Lesieur, Metais and Comte [12].

The idea is simple: if small \( \delta \) is the mesh cell width, then instead of calculating velocity in the form of \( \mathbf{V}(x, t) \) in the center point, an average form of velocity for that mesh, \( \bar{V}(x, t) \), will be calculated. For example, a small cell with an equal small length of \( \delta \) in location \( x(x_1, x_2, x_3) \) will have the average of

\[
\bar{V}(x, t) = \frac{1}{\delta} \int_{x_1 - \delta/2}^{x_1 + \delta/2} \int_{x_2 - \delta/2}^{x_2 + \delta/2} \int_{x_3 - \delta/2}^{x_3 + \delta/2} \mathbf{V}(y_1, y_2, y_3, t) \, dy_1 \, dy_2 \, dy_3 \quad (19)
\]
Thus, a filter function of \( \delta^{-3} g(\frac{x}{\delta}) \) is

\[
g(x) = \begin{cases} 
1 & \text{for } |x_j| \leq \frac{1}{\bar{z}} \\
0 & \text{for else}
\end{cases} \quad (20)
\]

Other filters are also needed to generate an LES solver [13].

DES, or hybrid RANS/LES, was illustrated by Spalart et al. [14] in 1997 to ease the numerical difficulties of LES and present a more accurate result than RANS. The wall regions were treated in a RANS-like manner, and the rest was solved as an LES problem. Here, they replaced the distance function \( (d) \), by \( \tilde{d} \), where

\[
\tilde{d} = \min[d, C_{DES}\Delta] \quad (21)
\]

where \( C_{DES} \) is a constant and equal to 0.65, and \( \Delta \) is the largest dimension of the grid cell between \( \Delta x, \Delta y, \) and \( \Delta z \). Spalart also presented a guide for the DES model grid, which is also available [15].

Other DES models that use the realizable \( k-\varepsilon \) and shear stress transport (SST) \( k-\omega \) models in the RANS (near wall) region are also available [16].

In 2006, Spalart et al. [17] introduced another numerical simulation approach, which was intended to ease the transition between the RANS and LES calculation region. In their model, the reference length was re-modified by replacing the distance function to

\[
\tilde{d} \equiv d - f_d \max(0, d - C_{DES}\Delta) \quad (22)
\]

where

\[
f_d \equiv 1 - \tanh ([8r_d]^3) \quad (23)
\]

and

\[
r_d \equiv \frac{\nu + \nu_t}{\sqrt{\nu_t} \sqrt{\nu} \kappa^2 \delta^2} \quad (24)
\]
where \( \nu_t \) is kinematic eddy viscosity, \( \nu \) is molecular viscosity, \( U_{ij} \) is the velocity gradients, \( \kappa \) is the von Karman constant, and \( d \) is distance to the wall. The terms \( C_{DES} \) and \( \Delta \) are the same as DES. If \( f_d = 0 \), the delayed detached eddy simulation (DDES) would be like a Spalart-Allmaras-Reynolds average Navier-Stokes (SA for RANS) model, and if \( f_d = 1 \), then DDES would be like a DES model.

DNS is one the oldest numerical solutions. In 1916, Lamb talked about the difficulties of DNS for NS equations, and in 1922 Richardson proposed a DNS scheme for an atmospheric simulation [18], [12]. Even today, DNS is used for a small Reynolds flow and very simple geometries. The three-dimensional (3D) solutions in these conditions are also fairly recent. The number of cells required for DNS cases is very large. For example, in a channel with height \( H \) and using uniform spacing, the grid number is \((100Re_t)^{\frac{9}{4}}\); reducing that number to achieve the minimum number of grids (i.e., reducing the cells not needed to be too small), the cell number will become \((3Re_t)^{\frac{9}{4}}\). This comes from the Kolmogorov time scale, which equals \( \left( \frac{\nu}{\epsilon} \right)^{\frac{4}{3}} \). Moin and Moser showed that the required time step for a channel is \( \Delta t = \frac{0.003H}{u_{*} \sqrt{Re_t}} \) [8].

In conclusion relative to turbulent models, because of current restrictions on processor performance, DES and DDES models prove to be more convenient, since DNS computation is costly (time and energy), and as mentioned by Catalano et al. [19], the near wall region in LES needs to be treated like a DNS grid system. Major load on flow solving in LES is actually in the near-wall (boundary layer) region [20].

1.2 Past Work

From the Wright Brothers’ first flight in 1903 until today, aerospace engineering has undergone major achievements and advances. This has also affected the field of fluid dynamics
and, in particular, aerodynamics. Research on SWBLI and SWTBLI has advanced in terms of simulation correction, run time, solution speed, and many other issues. Some research has resulted from the advancement in wind tunnels and associated facilities; other research is the result of the perfection of numerical techniques and computation.

1.2.1 Historical Research

It has been said that the first observation of a shock wave boundary layer interaction was made by Ferri in 1939 [21]. He presented experimental results of tested airfoils in a high-speed wind tunnel and discussed the difference between available analytical solutions and those of his experiments [22]. Pioneering experiments in this field performed by Liepmann in 1946, Ackeret, Feldman, and Rott in 1946, and Liepmann, Roshko, and Dhawan in 1951, were focused on transonic interactions, so that supersonic flight could be possible [23]. In 1955, the Aeronautical Research Council of the British Ministry of Supplies published a report, “The Interaction Between Shockwaves and Boundary Layers,” written by Holder et al. [24]. Here, the authors state that “The differences between the interaction with laminar and turbulent boundary layers are often a source of serious discrepancy between model experiments and full-scale conditions. For small-scale models it is, therefore, frequently essential to make the boundary layer turbulent by artificial means” [24]. In 1963, Arens and Spiegler [25] published “Shock-Induced Boundary Layer Separation in Overexpanded Conical Exhaust Nozzles,” Bennett [26] investigated “Shock-Wave Boundary-Layer Interaction on a Missile Nose Probe,” and Rubin [27] published “Shock Curvature Effect on the Outer Edge Conditions of a Laminar Boundary Layer.” A more complete history of shock wave boundary layer interaction and developments in the first 50 years of its research (up to year 2000) was performed by Dolling [21]. Another review on this topic is
presented by Andreopoulos et al. [28]. Reviews on recent works have also been published by Edwards [29] and Zheltovodov [30]. Figure 1 illustrates a schematically SWBLI occurrence.

![Figure 1. Shock wave turbulent boundary interaction scheme [31].](image)

1.2.2 Numerical Research

Table 2 lists chronologically the most recent and significant numerical research in SWBLI and SWTBLI.

**TABLE 2**

NUMERICAL INVESTIGATIONS OF SWBLI AND SWTBLI

<table>
<thead>
<tr>
<th>Researcher(s)</th>
<th>Mach</th>
<th>Scheme</th>
<th>Object</th>
<th>Year</th>
</tr>
</thead>
<tbody>
<tr>
<td>Travin et al. [33]</td>
<td>( Re = 1.4 \times 10^5 ), ( 3 \times 10^6 )</td>
<td>DES-SA</td>
<td>Circular cylinder</td>
<td>1999</td>
</tr>
<tr>
<td>Rohde [34]</td>
<td>0.5</td>
<td>ROE scheme</td>
<td>Ellipsoidal disk</td>
<td>2000</td>
</tr>
<tr>
<td>Barakos and Drikakis [35]</td>
<td>0.875</td>
<td>Two- and three-equation, eddy-viscosity turbulence models</td>
<td>Model validation over a bump</td>
<td>2000</td>
</tr>
<tr>
<td>Serre et al. [36]</td>
<td>( Re = 4000 )</td>
<td>DNS</td>
<td>Wall bounded rotating flow</td>
<td>2001</td>
</tr>
<tr>
<td>Moss [37]</td>
<td>9.3–11.4</td>
<td>Direct simulation Monte Carlo method of Bird</td>
<td>Hollow cylinder flares and double cones</td>
<td>2001</td>
</tr>
<tr>
<td>Kyle et al. [38]</td>
<td>0.676–0.750</td>
<td>Johnson-King non-equilibrium half-equation and Baldwin-Lomax algebraic</td>
<td>RAE 2822 and CAST 7 airfoils</td>
<td>2001</td>
</tr>
<tr>
<td>Hasan and McGuirk [39]</td>
<td>0.875</td>
<td>Various LES and k-( \varepsilon ) numerical and experimental models</td>
<td>10% axisymmetric bump</td>
<td>2001</td>
</tr>
<tr>
<td>Garnier et al. [40]</td>
<td>2.3</td>
<td>LES</td>
<td>Flat plate with oblique shock</td>
<td>2002</td>
</tr>
<tr>
<td>Lawal and Sandham [41]</td>
<td>1</td>
<td>DNS</td>
<td>Bump in channel</td>
<td>2002</td>
</tr>
<tr>
<td>Researcher(s)</td>
<td>Mach</td>
<td>Scheme</td>
<td>Object</td>
<td>Year</td>
</tr>
<tr>
<td>------------------------</td>
<td>------</td>
<td>----------------------</td>
<td>-----------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>Catalano et al. [19]</td>
<td>$Re_D = 10^6$</td>
<td>LES</td>
<td>Circular cylinder</td>
<td>2003</td>
</tr>
<tr>
<td>Bron [42]</td>
<td>0.641–0.702 and 0.549–0.688</td>
<td>$k-\omega$</td>
<td>2D and 3D bump</td>
<td>2004</td>
</tr>
<tr>
<td>Gerolymos et al. [43]</td>
<td>2.9</td>
<td>FANS</td>
<td>Oblique SWTBLI</td>
<td>2004</td>
</tr>
<tr>
<td>Loginov et al. [44]</td>
<td>2.95</td>
<td>LES</td>
<td>Compression corner</td>
<td>2004</td>
</tr>
<tr>
<td>Teramoto [45]</td>
<td>2.3</td>
<td>LES</td>
<td>Transitional BL</td>
<td>2005</td>
</tr>
<tr>
<td>Moroianu et al. [46]</td>
<td>0.6</td>
<td>LES</td>
<td>Bump in channel</td>
<td>2005</td>
</tr>
<tr>
<td>Bhanderi and Babinsky [47]</td>
<td>1.33, 0.7, 0.875</td>
<td>RANS with laminar viscous stresses, Reynolds normal stress and shear stress and adopting Morkovin’s hypothesis</td>
<td>Flat plate, bumps</td>
<td>2005</td>
</tr>
<tr>
<td>Wong et al. [48]</td>
<td>1.29</td>
<td>Baldwin-Lomax and $k-\omega$</td>
<td>Flat plate, 3D ramp bump in wind tunnel</td>
<td>2005</td>
</tr>
<tr>
<td>Wollbald et al. [49]</td>
<td>1.27</td>
<td>LES</td>
<td>Bump</td>
<td>2006</td>
</tr>
<tr>
<td>Alfano et al. [51]</td>
<td>0.73</td>
<td>Weakly non-linear correction and partially-averaged Navier-Stokes equations (PANS) approach applied to standard Launder and Sharma $k-\varepsilon$</td>
<td>External flows over a supercritical airfoil in buffet regime and transonic flows in Sajben nozzle</td>
<td>2006</td>
</tr>
<tr>
<td>Reinartz et al. [52]</td>
<td>8.3, 7.42</td>
<td>FLOWer and CFD++ Commercial Codes</td>
<td>Supersonic inlet</td>
<td>2006</td>
</tr>
<tr>
<td>Pino Martin et al. [53]</td>
<td>3</td>
<td>DNS</td>
<td>Compression Corner</td>
<td>2006</td>
</tr>
<tr>
<td>Balakumar [54]</td>
<td>1.8</td>
<td>5th-order accurate weighted essentially non-oscillatory (WENO) scheme and third-order total-variation diminishing (TVD) Runge-Kutta</td>
<td>BL control using suction</td>
<td>2006</td>
</tr>
<tr>
<td>Visbal et al. [55]</td>
<td>$Re_H=65,000$</td>
<td>$k-\varepsilon$ and Implicit LES</td>
<td>3D axisymmetric bump</td>
<td>2007</td>
</tr>
<tr>
<td>Oliver et al. [56]</td>
<td>2.87, 2.94, 11</td>
<td>SA and Menter’s SST and Olsen and Coakley lag mode ($k-\omega$) in commercial code: OVERFLOW</td>
<td>Compression corners, axisymmetric cone-flare</td>
<td>2007</td>
</tr>
<tr>
<td>Oliver et al. [57]</td>
<td>2.87, 2.94</td>
<td>SA, SST, and $k-\omega$ in Commercial code: OVERFLOW</td>
<td>Compression corners</td>
<td>2008</td>
</tr>
<tr>
<td>Vallet [58]</td>
<td>2.25</td>
<td>Low-Reynolds, standard $k-\varepsilon$</td>
<td>Compression corners</td>
<td>2008</td>
</tr>
<tr>
<td>Bodin and Fuchs [59]</td>
<td>1.1</td>
<td>LES</td>
<td>Bump in channel</td>
<td>2008</td>
</tr>
<tr>
<td>Barakos et al. [60]</td>
<td>0.783</td>
<td>URANS $k-\omega$ and LES</td>
<td>10% axisymmetric bump</td>
<td>2008</td>
</tr>
<tr>
<td>Tanaki et al. [61]</td>
<td>1.5, 1.9, 3, 5</td>
<td>Launder-Sharma low-Reynolds $k-\varepsilon$</td>
<td>Micro-channel</td>
<td>2009</td>
</tr>
</tbody>
</table>
1.2.3  Experimental Research

Table 3 lists chronologically the most recent and significant experimental research in SWBLI and SWTBLI done in a wind tunnel.

**TABLE 3**

EXPERIMENTAL INVESTIGATIONS OF SWBLI AND SWTBLI

<table>
<thead>
<tr>
<th>Researcher(s)</th>
<th>Mach</th>
<th>Facility</th>
<th>Object</th>
<th>Year</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liu and Squire [63]</td>
<td>1-1.82</td>
<td>Blow-down supersonic wind tunnel, University of Cambridge</td>
<td>2D bump</td>
<td>1988</td>
</tr>
<tr>
<td>Kimmel and Poggie [64]</td>
<td>6</td>
<td>Arnold Engineering Development Center, vonKarman Gas Dynamics Facility Tunnel B</td>
<td>Sharp nosed cone with half-angle of 7º</td>
<td>2000</td>
</tr>
<tr>
<td>Reijasse et al. [65]</td>
<td>1.95</td>
<td>ONERA research facilities of Chalais-Meudon Center</td>
<td>Hypersonic engine inlet</td>
<td>2001</td>
</tr>
<tr>
<td>Schulte et al. [66]</td>
<td>6</td>
<td>German Aerospace Research Center hypersonic blow-down wind-tunnel facility</td>
<td>Hypersonic engine inlet</td>
<td>2001</td>
</tr>
<tr>
<td>Gefroh et al. [67]</td>
<td>2.41</td>
<td>Gas Dynamics Laboratory, University of Illinois at Urbana-Champaign</td>
<td>SWBLI control using mesoflaps</td>
<td>2002</td>
</tr>
<tr>
<td>Sigfrids [68]</td>
<td>0.69</td>
<td>Department of Energy Technology, Royal Institute of Technology (KTH)</td>
<td>2D bump</td>
<td>2003</td>
</tr>
<tr>
<td>Prioris and Babinsky [69]</td>
<td>1.22, 1.32, 1.48</td>
<td>Blow-down supersonic wind tunnel, University of Cambridge</td>
<td>Circular arc bump</td>
<td>2003</td>
</tr>
<tr>
<td>Holden et al. [70]</td>
<td>10-12</td>
<td>Calspan-UB Research Center (CUBRC)</td>
<td>Hollow cylinder/flare and double cone configurations</td>
<td>2003</td>
</tr>
<tr>
<td>Holden and Babinsky [71]</td>
<td>1.3, 1.5</td>
<td>Supersonic wind tunnel, University of Cambridge</td>
<td>SWBLI control using cavity, 3D bumps, slots and datum grooves</td>
<td>2003</td>
</tr>
<tr>
<td>Bueno et al. [72]</td>
<td>2</td>
<td>Blow-down wind tunnel, University of Texas at Austin</td>
<td>Upstream mass injection on turbulent</td>
<td>2004</td>
</tr>
<tr>
<td>Bron [42]</td>
<td>0.641-0.702 and 0.549-0.688</td>
<td>Department of Energy Technology, Royal Institute of Technology (KTH)</td>
<td>2D and 3D bump</td>
<td>2004</td>
</tr>
<tr>
<td>Szwaba [73]</td>
<td>1.35, 1.45, 1.55</td>
<td>IMP PAM transonic wind tunnel</td>
<td>SWBLI control with air jet vortex generator</td>
<td>2005</td>
</tr>
<tr>
<td>Researcher(s)</td>
<td>Mach</td>
<td>Facility</td>
<td>Object</td>
<td>Year</td>
</tr>
<tr>
<td>---------------</td>
<td>------</td>
<td>----------</td>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>Tillotson et al. [74]</td>
<td>2.95</td>
<td>Blow-down wind tunnel, Gas Dynamics Laboratory, University of Illinois</td>
<td>Bump compression surface</td>
<td>2006</td>
</tr>
<tr>
<td>Schulein [75]</td>
<td>5</td>
<td>DLR Gottingen, Ludwig tube facility (RWG)</td>
<td>Skin-friction and heat flux measurement of flat plate and oblique shock SWTBLI</td>
<td>2006</td>
</tr>
<tr>
<td>Ogawa and Babinsky [76]</td>
<td>1.3, 1.5</td>
<td>Novel supersonic wind tunnel, University of Cambridge</td>
<td>Control of normal shock from bump</td>
<td>2006</td>
</tr>
<tr>
<td>Murphree et al. [77]</td>
<td>5</td>
<td>Blow-down wind tunnel, University of Texas at Austin</td>
<td>Transitional BL</td>
<td>2006</td>
</tr>
<tr>
<td>Ganapathisubramani, et al. [78]</td>
<td>2</td>
<td>Blow-down wind tunnel, University of Texas at Austin</td>
<td>Large-scale coherence characteristics</td>
<td>2007</td>
</tr>
<tr>
<td>Murphree et al. [79]</td>
<td>5</td>
<td>Blow-down wind tunnel, University of Texas at Austin</td>
<td>Transitional no-trip BL</td>
<td>2007</td>
</tr>
<tr>
<td>Ogawa et al. [80]</td>
<td>1.3</td>
<td>Blow-down supersonic wind tunnel, University of Cambridge</td>
<td>SWBLI control using 3D bump</td>
<td>2007</td>
</tr>
<tr>
<td>Bruce and Babinsky [81]</td>
<td>1.4, 1.5</td>
<td>Blow-down supersonic wind tunnel, University of Cambridge</td>
<td>Unsteady periodic forcing in a parallel walled duct</td>
<td>2008</td>
</tr>
<tr>
<td>Ogawa and Babinsky [82]</td>
<td>1.5</td>
<td>Blow-down supersonic wind tunnel, University of Cambridge</td>
<td>SWBLI control using 3D bump</td>
<td>2008</td>
</tr>
<tr>
<td>Barakos et al. [60]</td>
<td>0.783</td>
<td>In-draft transonic wind tunnel, Queen’s University of Belfast</td>
<td>10% axisymmetric bump</td>
<td>2008</td>
</tr>
<tr>
<td>Bruce and Babinsky [83]</td>
<td>1.5</td>
<td>Blow-down supersonic wind tunnel, University of Cambridge</td>
<td>SWBLI control using micro-vane vortex generator</td>
<td>2009</td>
</tr>
<tr>
<td>Polivanov et al. [84]</td>
<td>2</td>
<td>T-325 of Institute of Theoretical and Applied Mechanics (ITAM)</td>
<td>Flat plate</td>
<td>2009</td>
</tr>
<tr>
<td>Sahoo et al. [85]</td>
<td>7.2</td>
<td>Mach 8 hypersonic boundary layer, Princeton Gas Dynamics Laboratory</td>
<td>Surface transportation effect in low Reynolds</td>
<td>2009</td>
</tr>
<tr>
<td>Holden et al. [86]</td>
<td>7-14</td>
<td>CUBRC shock tunnel</td>
<td>BL types over axisymmetric and elliptic cones</td>
<td>2009</td>
</tr>
<tr>
<td>Herges et al. [87]</td>
<td>1.4</td>
<td>Blow-down wind tunnel, University of Illinois at Urbana-Champaign</td>
<td>Micro-ramp</td>
<td>2009</td>
</tr>
</tbody>
</table>

### 1.3 Computational Software

Simulations were performed using the following commercial software: Ansys-Fluent [88] and Cobalt [89]. Both software have the capability of simulating hybrid RANS/LES (DES) and DDES models.
CHAPTER 2

MESH GENERATION

In computation fluid dynamics, mesh in two dimensions, or a plane or surface mesh, is either structured or unstructured. Structured mesh is a mesh whereby the elements are square or each four points generate a calculating element. Implied in the terminology is that the mesh should have equal points on facing boundaries in a four-line bounded domain (surface). Unstructured mesh uses three elements and creates a triangular calculating element. Unlike structured mesh, unstructured mesh is generated in various ways. It starts from nodes (or grid points) from boundary lines and follows the guideline implemented in the mesh generation code/software and finalizes the mesh spanned on the surface. Basically, in this step, more points are created on the surface and off the boundary, and hence, the full surface is covered with triangular elements. Different ways for doing this may or may not consider the following: smallest allowed element size, maximum allowed relevant angle, boundary growth inside the surface (or dependence on boundary grid points), or simply the use of meshed guidelines. Since the focus in this thesis was on volume mesh (3D object), this case will be discussed in depth.

2.1 Volume Mesh

Because of the different commercial software used to generate mesh, there are different mesh-generation procedures or options. Each software and its volume mesh capabilities are discussed in the next two sections.

2.1.1 Gambit

The grid-generator Gambit [90] has the option of creating elements in various shapes: hexagonal, hybrid hexagonal/wedge, and hybrid tetrahedral/hexagonal, pyramidal or wedge [91].
As shown in Figures 2 to 5, Gambit has the ability to create different volume elements. The shapes generated are not restricted to a fixed number of nodes; therefore, the number of nodes varies. This is very convenient because some points are set as guidelines in generating the volume mesh elements.

Figure 2. Hexahedron volume mesh patterns: (a) 8 nodes, (b) 20 nodes, and (c) 27 nodes [91].

Figure 3. Wedge volume mesh patterns: (a) 6 nodes, (b) 15 nodes, and (c) 18 nodes [91].
2.1.2 Gridgen

Grid-generation software Gridgen and Pointwise [92] have the ability to create structured hexagonal mesh using the following techniques: LaPlace or fixed grid (smoothness), Thomas-Middlecoff (clustering), and von Lavante-Hilgenstock-White or Steger-Sorenson (orthogonality) [93]. For unstructured mesh, this software uses tetrahedral elements with a technique introduced
by Steinbrenner and Abelanet called T-Rex [94]. This software also has the ability to make three
tetrahedrals into one prism for reducing cell count. Direct Prism is also available to create prism
elements from surface nodes. Another technique to create tetrahedral mesh volumes is by using a
modified Delaunay method. Post-grid generation improvement is available by edge swapping
and LaPlacian smoothing [95].

2.2 General DES Mesh Condition

Mesh for creating a computational domain acceptable for DES solutions should be
generated using Spalart’s guide for generating acceptable mesh for a DES solver [15] and current
advancements in computing facilities and standards for acceptable simulations. Total geometry is
separated into three major sections: Euler region (ER), RANS region, and LES region. The ER
runs from upstream to the near boundary layer or wall segment. This section does not require
heavy grid clustering and grid focus, and its main purpose is to develop the flow to a turbulent
condition, or to be in a far position so that downstream disturbances do not affect the input
(inlet). In the guideline provided, first meshes could even be of a size greater than the object’s
diameter. It is recommended by software manuals that the far field region be in a location that
has a diameter of at least ten times the diameter of the object. With a larger grid size comes a
simpler solution algorithm (assuming Euler is easier than RANS, which is easier than LES).
Other regions in the computational domain should be treated as an ER, if and only if they do not
have flow disturbances.

The near-wall region is called the RANS region. In this region, the general rule of
meshing is to satisfy the following:

- The number of grid cells in the boundary layer should be approximately 20 and not less
  than 15.
• \( y^+ \) for the first grid cell \((y_1^+)\) should be less than 1.

• Grid clustering should be in near agreement with this spacing: \( \Delta y_{i+1}/\Delta y_i \approx 1.25 \) and

\[
\frac{\Delta y_{i+1}^+}{\Delta y_i^+} \approx 1.3
\]

• In sections where flow is emerging from the boundary layer toward the free stream (not wall bounded), the following could be used: \( \Delta y_i/y_i \approx \log(1.25) \).

• In locations further from the wall than \( \frac{\delta}{2} \) (where \( \delta \) is the boundary layer thickness), the grid should be generated as \( \Delta y_i \approx \delta \log(1.25) \).

Spalart reported that a finer mesh, where \( \Delta y_i^+ < 1 \) and \( \Delta y_{i+1}/\Delta y_i < 1.2 \), has less effect on the solution accuracy [15].

The region in which large eddies are present, and thus there is flow separation, is called the LES region. For this region, mesh is required to be finer than the other two regions. Turbulence and vorticity in this region is not caused by the wall boundary layer, and it is not aligned with a grid spanning along the thin-wall shear layers. The LES region is separated into two sub-regions: focus region (FR) and departure region (DR). Between the RANS and LES regions, a thin section called the viscous region exists. The RANS viscous and LES viscous regions (in DES, the RANS or LES regions are made according to the solver, due to the modified wall distance) could be treated in the same way. In DDES, the transition between RANS and LES is done more smoothly; thus, the boundary entities are more affected in the LES region. Spanning, for example in a channel, could be as great as \( \Delta x^+ = \Delta z^+ = 8,000 \) [15]. For simplicity, it is assumed that both LES and RANS viscous regions are treated as the RANS viscous region, not the LES viscous region.
The main purpose of the departure region is to assure that the flow simulated in the FR is not disturbed. The grid spacing between the FR and the exit flow (outflow or far field, etc.) that lies in the DR will not be used. So in this case, grid spacing is not as critical in the DR as it is in the FR. A valid cutoff section must be kept in mind when developing the computational domain. The main issue in the DR is that grid quality should satisfy the guideline on flow solvers, i.e., maximum grid skewness and maximum grid aspect ratio. This is also the primary issue in the ER.

2.3 Object and Computational Domain

Dimensions of the two objects to be meshed are described in the following sections. The bump was spanned in the Y direction, so only the X-Z coordination were required in order to generate the full 3D mesh. The antenna with the given dimensions was placed on a flat surface. The dimension of the lower plate should be in agreement with both the solver requirement and the actual object configuration.

2.3.1 Bump

The first object to be meshed was a bump that was simulated in a wind tunnel-like domain. The coordination of the bump (in millimeters) is shown in Table 4. Figure 6 shows a cross section of the bump, and as shown in Figure 7, the full span domain is a 3D element. Figure 8 shows the computational domain 3D mesh. This mesh was validated by 2D results with a different mesh density.

The total domain length \( (l) \) was 29 cm, height \( (z_{max}) \) was 12 cm, and test section width was 10 cm. To make the simulation similar to a wind-tunnel experiment, it was assumed that the boundary layer is generated prior to the flow entrance. For this case, a boundary layer was generated from a 50 cm-long duct.
<table>
<thead>
<tr>
<th>X</th>
<th>Z</th>
<th>X</th>
<th>Z</th>
<th>X</th>
<th>Z</th>
<th>X</th>
<th>Z</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-70.00</td>
<td>0.00</td>
<td>14.33</td>
<td>2.53</td>
<td>37.90</td>
<td>9.97</td>
<td>61.11</td>
<td>9.82</td>
<td>84.33</td>
<td>6.79</td>
</tr>
<tr>
<td>-56.51</td>
<td>0.00</td>
<td>17.58</td>
<td>3.89</td>
<td>40.80</td>
<td>10.24</td>
<td>64.04</td>
<td>9.56</td>
<td>87.23</td>
<td>6.32</td>
</tr>
<tr>
<td>-43.11</td>
<td>0.00</td>
<td>20.48</td>
<td>5.18</td>
<td>43.70</td>
<td>10.40</td>
<td>66.92</td>
<td>9.25</td>
<td>90.13</td>
<td>5.85</td>
</tr>
<tr>
<td>-29.72</td>
<td>0.00</td>
<td>23.39</td>
<td>6.41</td>
<td>46.60</td>
<td>10.47</td>
<td>69.82</td>
<td>8.91</td>
<td>93.04</td>
<td>5.38</td>
</tr>
<tr>
<td>-16.39</td>
<td>0.00</td>
<td>26.29</td>
<td>7.50</td>
<td>49.50</td>
<td>10.46</td>
<td>72.72</td>
<td>8.54</td>
<td>95.94</td>
<td>4.92</td>
</tr>
<tr>
<td>-5.13</td>
<td>0.00</td>
<td>29.19</td>
<td>8.39</td>
<td>52.41</td>
<td>10.38</td>
<td>75.62</td>
<td>8.13</td>
<td>98.84</td>
<td>4.47</td>
</tr>
<tr>
<td>3.45</td>
<td>0.06</td>
<td>32.09</td>
<td>9.07</td>
<td>55.31</td>
<td>10.24</td>
<td>78.52</td>
<td>7.70</td>
<td>101.96</td>
<td>4.01</td>
</tr>
<tr>
<td>9.78</td>
<td>1.01</td>
<td>34.99</td>
<td>9.59</td>
<td>58.21</td>
<td>10.05</td>
<td>81.43</td>
<td>7.25</td>
<td>107.25</td>
<td>3.29</td>
</tr>
</tbody>
</table>

Figure 6. Bump view from a plane slice.

Figure 7. Full bump model for wind tunnel [42].
Clustering in the near-wall regions was performed, with 20 grid points in the boundary layer near the lower wall and 15 grid points near the other sides. For this simulation, the first grid from the lower surface was positioned so that $y^+ < 1$.

To ensure that $y^+$ would meet the standards for a DES grid system, the following equations were derived:

$$y^+ \equiv \frac{u^+ y}{v} \quad (25)$$
\[ u_+ \equiv \sqrt{\frac{\tau_w}{\rho}} \]  
\[ \tau_w = \left( \frac{\partial u}{\partial y} \right)_{y=0} \]  

where \( \nu \) is kinematic viscosity, and \( \rho \) is density. Substituting these equations to create a relation between \( y^+ \) and \( y_1 \) (first grid cell from the surface) yields

\[ y^+ \equiv \frac{y_1}{\nu} \sqrt{\frac{u_1}{\rho y_1}} \]  

Since \( u = u_\infty f(\eta) \) and \( y = \eta \frac{\nu x_1}{u_\infty} \), then

\[ y^+ \equiv \frac{\eta_1}{\nu} \sqrt{\frac{u_\infty f(\eta_1)}{\rho \eta_1 \frac{\nu x_1}{u_\infty}}} \equiv \frac{\eta_1 x_1 f(\eta_1)}{\nu \rho \frac{\nu x_1}{u_\infty}} \rightarrow y^+ \equiv \sqrt{\frac{\eta_1 f(\eta_1) \sqrt{Re_x}}{\mu}} \]

where \( x \) in \( Re_x \) is the distance that the boundary layer was developed, and \( \mu \) is the dynamic viscosity. Also, this equation provides a good assessment for \( y_1 \), however due to the many assumptions in development of the equation, it may not always produce an accurate prediction (although the answer is always close). Therefore, the grid needs to be modified in order to find the correct \( y_1 \) so that \( y^+ < 1 \).

A 2D model was first generated to identify the shock location and solution procedure. Since simulating a wind tunnel needs a full 3D computational domain for taking into consideration the wall effect, a 3D mesh was generated with a total of 924,300 structured hexahedral volume cells. The 2D and 3D bump meshes were generated by Gambit.

### 2.3.2 Antenna

The second object to be meshed was a design for an aircraft UHF antenna, similar to a DM-C34 series airborne UHF SATCOM antenna, as shown in Figure 9, which is standard in many aircraft carriers, such as the C-130 Hercules, C-141 Starlifter, and C-5 Galaxy.
This antenna design is a circular disk with curved edges. The disk diameter in the main section is 49 cm, and the disk thickness is 4 cm. The disk has a curvature of radius of 2 cm on its edges. Figure 10 shows a top view of this antenna.

The disk would be mounted on an elliptical frustum. The base ellipse has a major axis of 37 cm and minor axis of 17 cm. The frustum’s height is 26 cm, and the upper ellipse has major and minor axes of $29 \times 14$ cm, as shown in Figure 11.
The computational domain of the antenna, as shown in Figure 12, is divided into five sections: three are ER, then the domain in which antenna is located is primarily RR, and the LR is far from the surface and the domain behind it is the LR.

The computational domain is nearly 10D in length (D is the largest diameter of the object, i.e., center disk) and 3H (H is the antenna’s height) in height. The mesh was initially generated with Gridgen, and final corrections were made with Pointwise. The mesh near the
antenna was denser, as illustrated in a side view in Figure 13. A total of 2.8 million unstructured tetrahedral grid volumes were generated for this case. Figure 14 shows the surface mesh near the antenna protrusion.

Figure 13. Mesh around antenna with a plane of $y = 0$ slice.

Figure 14. Surface mesh near antenna protrusion.
CHAPTER 3
SOLUTION PROCEDURE

This chapter describes how the two protrusion’s computational domain (mesh) was generated. Each object has its own type of volume mesh. The generated mesh type could be applied to objects that have a similar configuration.

3.1 Bump

The bump protrusion mesh was developed in Gambit, the commercial software Fluent was used for the flow simulation. Inflow was selected to be air in an ideal gas form. Viscosity was calculated with Sutherland’s two-equation formula. The inlet was set as the pressure inlet.

Since the results were compared with an experimental wind-tunnel simulation, the computation parameters were chosen to be in agreement with the experiment configuration. One parameter was the boundary layer thickness before the protrusion. For similarity, the computational domain could be extended (add a 50 cm duct), or the boundary layer could be calculated (generated) and provided as input. To create the boundary layer in this case, the following was used:

\[ P_t = P_s + q(d) \]  

where

\[
\begin{align*}
    u &= u(\delta) \quad \text{if } d < \delta \\
    u &= u_\infty \quad \text{if } d \geq \delta
\end{align*}
\]  

(30)

\[ q(d) = \frac{1}{2} \rho (V(d))^2 \]  

(31)

\[ 0 \leq V(d) \leq V_\infty \rightarrow P_s \leq P_t \leq P_s + q \]  

(32)

The gage pressure was 120 kPa, and the total pressure was varied from 120 kPa to 160 kPa in the log region (or boundary layer) and was kept constant at 160 kPa everywhere else (typical boundary layer is shown in Figure 15). For current simulation, a 3D boundary layer was
generated for the pressure inlet inflow. The outlet static pressure was 106 kPa, so $M_\infty = 0.702$.

The side boundaries were all adiabatic, with no slip walls and a temperature of 303 K.

![Figure 15. Schematic of boundary layer.](image)

The solver was set to be 3D, unsteady, implicit formulation with the Green-Gauss cell-based gradient option. The unsteady formulation was a second-order implicit, and the porous formulation was superficial velocity. Using an axisymmetric solver was not allowed with the DES model; thus, the full domain was modeled.

Simulation was executed by using the semi-implicit method for pressure-linked equations (SIMPLE) method for pressure-velocity coupling and a second-order upwind scheme for pressure, density, and modified turbulent viscosity. Energy discretization and momentum used bounded central difference. As is known, the second-order upwind scheme is less diffusive than the first-order upwind scheme, although convergence occurs faster in the first-order upwind scheme, but flow is not always parallel to the grid pattern, especially in the near-wall region.
Therefore, as recommended by the solver, a second-order upwind scheme was used for these cases.

SIMPLE and SIMPLEC (the semi-implicit method for pressure-linked equations, consistent) are recommended for cases that involve LES models, and SIMPLEC is recommended for domains that have highly skewed mesh (maximum skewness available for Fluent to start simulation is 0.98); therefore, the SIMPLE method was chosen.

For faster solution convergence, a time-advancement unsteady DES-SA solver with sub-iterations was used. Flow conditions for inflow and outflow were unchanged, in order to create a steady-state flow condition. Solution time advancement is when the difference of all variables is less than $10^{-4}$. The solution for the first millisecond (0.001 s) uses a time step of $10^{-6}$ and a maximum of 50 sub-iterations (after a few time steps, the cutoff point is about 10–20 sub-iterations). The solution after that uses a time step of $10^{-5}$, up to 1 second, with a maximum of five sub-iterations.

For this simulation, high-accuracy models were used so that these procedures could be validated and thus suggested for basically all DES-SA models in Fluent.

3.2 Antenna

The antenna’s mesh was developed in Gridgen. The computational domain was an installed antenna on a flat plane. This configuration was chosen in order to assist the flow parameters without any disturbances other than the protrusion. Commercial software Fluent and Cobalt were used for the flow simulations. The Fluent configuration was the same for both protrusion simulations, yet the boundary was changed into a wall boundary for only the antenna and lower plane (surface on which the antenna is installed), and farfield for everywhere else. Both solvers used a Mach number of 0.85 for ideal gas air.
Cobalt used a different solution requirement. The solver file (file that describes the requested solution procedure to the computer) was set to use turbulent NS model parameters, with the turbulence model of DDES-SA. The gas model was a perfect gas, flow was unsteady, spatial accuracy was second-order, and the solver used four Newton sub-iterations. Initially, the time step was set to $10^{-6}$, for a total of 1 millisecond (0.001 s), and after that, the time step was set to $10^{-5}$.

The solver calculated the flow properties with constants such as ratio of specific heats, gas constant, gravity, etc., and all values were set to the default value assumed by the Cobalt solver.

The farfield boundary used a modified Riemann invariant to calculate the surrounding flow properties. A modified Riemann invariant was suggested for the supersonic inflow, but this could also be used for other cases as well. The wall was set to be adiabatic with no slip, so the flow was assumed to have zero velocity in the first cell from the wall. A no-trips option was selected, because the flow input was far enough from the object, so the flow became turbulent prior to the protrusion. The farfield boundary was selected to satisfy the solver’s requirement and was in a reasonable distance, according to the three aircraft that have used SATCOM antennas.
CHAPTER 4
RESULTS

In this chapter, the simulation results are presented for two objects: bump and antenna. The antenna protrusion was simulated using DES and DDES numerical models. The antenna was also simulated with a DDES numerical model in a 10° side slip.

4.1 Bump

Bump results are discussed in the next three sections. The first section involves grid verification from the 2D solution. The second section shows the results compared with experimental and numerical results provided by Bron [42]. Also, $y^+$ justification is checked with the 3D result. The third section illustrates the flow pattern and fluid parameters from these simulations.

4.1.1 2D Solution

In order to estimate a correct number of mesh elements for the 3D bump, a 2D structured mesh was generated, and flow was simulated in the same manner as the 3D model. The solution procedure was the same as for the 3D bump, as described previously.

Figure 16 compares static pressure on the lower surface between the RANS-SA and experimental-numerical results from Bron [42]. Figure 17 shows $y^+$ on the lower surface. The bump was located on the lower surface; thus, the main focus was to have $y^+ < 1$ on the lower surface. Figure 18 shows the static pressure distribution in the computational domain. Figure 19 shows the Mach contours in the computational domain. To illustrate flow circulation and the turbulence in flow, Figure 20 was plotted to show turbulent viscosity, and Figure 21 was plotted to show velocity angle. As shown in Figure 20, the flow became turbulent a close distance from the inlet, and the overall flow around the bump was turbulent. Figure 21 shows the flow
circulation at the end of bump. Note that all figures showing the 2D bump are located on the X-Y plane, and flow enters from the left.

Figure 16. Flow on lower surface: (a) RANS-SA, (b) numerical and experimental data from Bron [42].
Figure 17. Bump $y^+$. 

Figure 18. Static pressure contours.
Figure 19. Mach contours.

Figure 20. Turbulent viscosity contours.
4.1.2 3D Bump Results Comparison

Simulating the bump in a wind tunnel revealed a phenomenon known as “wall effect.” Wall effect occurs to a symmetrical object (i.e., any section parallel to each other on a certain axis plane is identical) simulated in a wall-bounded environment. The results of a three-dimensional simulation vary for every section that is a different distance from the wall. Oliver et al. [57] investigated this phenomenon for a compression ramp. Results closer to the middle section of the computational domain (or closer away from the walls) were more similar to that of a 2D simulation. A full 3D domain was simulated, and results where compared with experimental and numerical results from the work of Bron [42].
Figure 22 compares static pressure on the lower surface, or 1 cm from the side wall. Results are compared with experimental data for DES-SA and $k-\omega$. Figures 23 and 24 are the same, except for distances of 25 mm and 50 mm from the side wall.

![Figure 22. Static pressure on lower surface comparison at 10 mm from side wall: (a) DES-SA and experimental data, (b) $k-\omega$ and experimental data from Bron [42].](image)

(a)

(b)
Figure 23. Static pressure on lower surface comparison at 25 mm from side wall: (a) DES-SA and experimental data, (b) $k-\omega$ and experimental data from Bron [42].
Figure 24. Static pressure on lower surface comparison at 50 mm from side wall: (a) DES-SA and experimental data, (b) $k-\omega$ and experimental data from Bron [42].
Comparing results from the numerical model with experimental data validates the approach in this thesis. The maximum error was less than 10% for static pressure value and 16% for shock location. The mean error was 5%–7% for these cases. Another comparison was conducted between DES-SA and $k$-$\omega$ numerical schemes. For some parts, the error of $k$-$\omega$ was less than that for DES-SA, but DES-SA managed to be more accurate in capturing the trend of pressure variation in the near-wall regions. DES-SA was also more accurate for pressure values and shock locations for the near-wall regions.

Another element investigated was the $y^+$ value. Figure 25 shows the spanwise $y^+$ value distribution for a lower surface. Figure 26 compares the $y^+$ value at 1 cm from the side wall and 5 cm from the side wall (middle section from the plane).

![Figure 25. Spanwise $y^+$ value distribution on the lower surface.](image)
Figure 26. $y^+$ value of lower surface compared with: (a) 1 cm from side wall, (b) 5 cm from side wall.
4.1.3 3D Bump Expanded Results

After the simulated results were validated with the experimental data, other flow properties were analyzed and presented. As mentioned previously, the wall effect was exposed by a difference in flow property values for both experimental and numerical simulations by considering the distance to the wall. As shown in Figure 27, streamlines in the middle of the computational domain are parallel to the lower surface from inflow to outflow. On the other hand, streamlines near the corners created a noticeably different pattern and were lifted from the parallel section near the lower wall and also created flow circulation. Figure 28 shows turbulent viscosity contours for distances of 1 cm, 2.5 cm, and 5 cm from the side wall.

Figure 27. Streamlines near lower surface with static pressure contours.
Turbulent Viscosity $\frac{Kg}{m.s}$

(a)

Turbulent Viscosity $\frac{Kg}{m.s}$

(b)
Figure 28. Turbulent viscosity contours: (a) 1 cm, (b) 2.5 cm, and (c) 5 cm from side wall.

Due to the flow circulation near the wall regions, as it approached the middle of the bump, the flow became less turbulent. Figure 29 shows the velocity angle of the flow for different distances from the wall. Note that in the 2D simulation, the bump is located in the X-Y plane, while in the 3D simulation, it is located in the X-Z plane. The flow-velocity angle near the middle section of the bump was mostly unchanged, similar to that in the 2D simulation. Again, the flow closer to the side walls rotated after the bump location; this is visualized in Figure 29 (a).
Figure 29. Velocity angle (deg): (a) 1 cm, (b) 2.5 cm, (c) 5 cm from side wall.

Figure 30 shows static pressure and Figure 31 shows Mach number contours in the computational domain at different locations from the side wall. The wall effect decreased shock intensity so that in the middle of the bump, the Mach number was at its peak. In the near-wall region, flow circulation created a decrease in the Mach number. Static pressure was quite similar for all regions of the computational domain except near the bump location, which was due to the shock wave.
Figure 30. Static pressure in the computational domain: (a) 1 cm, (b) 2.5 cm, and (c) 5 cm from side wall.
Figure 31. Mach contours in computational domain: (a) 1 cm, (b) 2.5 cm, and (c) 5 cm from side wall.
4.2 Antenna

Flow around the antenna was simulated using DES and DDES numerical models. Prior to the simulations, it was important to investigate these two numerical models. As mentioned previously, the DDES model delayed the switch point at which the RANS model was shifted to LES, so that the simulation was less affected by the quality of the grid, and boundary layer thickness could be achieved using a less dense grid near the wall. It was also important to investigate the shock turbulent boundary layer interaction around the antenna. Here, the DDES and DES models were used; conjointly, a DDES model with a side slip angle was also simulated. The DDES model was chosen based on its accuracy and the availability of computational resources.

Flow simulation also validated the accuracy of the computational domain structure by showing no divergence and a fine estimate of the upper flow path. Figure 32 shows that streamlines from the upper section of the antenna were not affected over half the maximum object height.

Figure 32. Streamlines and turbulent viscosity.
4.2.1 DES/DDES Antenna

Figures 33 to 62 show different flow properties (static pressure, temperature, turbulent viscosity) at various distances (10, 20, 26, 28, and 30 cm) from the lower surface. As can be seen in the object geometry, the first two are sections on the elliptic frustum stand, and the others are sections on the upper disk. Simulation using the DES model had a weaker boundary layer, and the shock location appeared onward of the location and further than expected for an ellipse with such an aspect ratio. On the other hand, the DDES model managed to predict shock location and boundary layer thickness around the elliptic frustum more discreetly. Each figure shows DDES and DES results for specified flow parameters at various locations. The results are shown in times steps of time in 0.15 second increments: $t = 0.15\ s$ and $t = 0.3\ s$.

![Figure 33](image1.png)

**Figure 33.** Static pressure at $Z = 10\ cm$, $t = 0.15\ s$: (a) DDES, (b) DES.

![Figure 34](image2.png)

**Figure 34.** Static pressure at $Z = 20\ cm$, $t = 0.15\ s$: (a) DDES, (b) DES.
Figure 35. Static pressure at $Z = 26$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 36. Static pressure at $Z = 28$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 37. Static pressure at $Z = 30$ cm, $t = 0.15$ s: (a) DDES, (b) DES.
Figure 38. Static pressure at $Z = 10$ cm, $t = 0.3$ s: (a) DDES, (b) DES.

Figure 39. Static pressure at $Z = 20$ cm, $t = 0.3$ s: (a) DDES, (b) DES.

Figure 40. Static pressure at $Z = 26$ cm, $t = 0.3$ s: (a) DDES, (b) DES.
Figure 41. Static pressure at $Z = 28 \text{ cm}$, $t = 0.3 \text{ s}$: (a) DDES, (b) DES.

Figure 42. Static pressure at $Z = 30 \text{ cm}$, $t = 0.3 \text{ s}$: (a) DDES, (b) DES.

Figure 43. Temperature at $Z = 10 \text{ cm}$, $t = 0.15 \text{ s}$: (a) DDES, (b) DES.
Figure 44. Temperature at $Z = 20$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 45. Temperature at $Z = 26$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 46. Temperature at $Z = 28$ cm, $t = 0.15$ s: (a) DDES, (b) DES.
Figure 47. Temperature at $Z = 30$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 48. Temperature at $Z = 10$ cm, $t = 0.3$ s: (a) DDES, (b) DES.

Figure 49. Temperature at $Z = 20$ cm, $t = 0.3$ s: (a) DDES, (b) DES.
Figure 50. Temperature at $Z = 26 \text{ cm}$, $t = 0.3 \text{ s}$: (a) DDES, (b) DES.

Figure 51. Temperature at $Z = 28 \text{ cm}$, $t = 0.3 \text{ s}$: (a) DDES, (b) DES.

Figure 52. Temperature at $Z = 30 \text{ cm}$, $t = 0.3 \text{ s}$: (a) DDES, (b) DES.
Figure 53. Turbulent viscosity at $Z = 10$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 54. Turbulent viscosity at $Z = 20$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 55. Turbulent viscosity at $Z = 26$ cm, $t = 0.15$ s: (a) DDES, (b) DES.
Figure 56. Turbulent viscosity at $Z = 28$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 57. Turbulent viscosity at $Z = 30$ cm, $t = 0.15$ s: (a) DDES, (b) DES.

Figure 58. Turbulent viscosity at $Z = 10$ cm, $t = 0.3$ s: (a) DDES, (b) DES.
Figure 59. Turbulent viscosity at $Z = 20$ cm, $t = 0.3$ s: (a) DDES, (b) DES.

Figure 60. Turbulent viscosity at $Z = 26$ cm, $t = 0.3$ s: (a) DDES, (b) DES.

Figure 61. Turbulent viscosity at $Z = 28$ cm, $t = 0.3$ s: (a) DDES, (b) DES.
Turbulent flow is considered unsteady, and turbulent properties and eddies vary in time. After a flow is fully simulated, the turbulent regions can be defined for acquiring time-dependent properties. A simple definition of turbulent flow is where the stream path (streamlines) tends to have a non-striate and twisted path. These paths also change in time.

Results from DDES simulation are used for flow study. Figure 63 shows a typical eddy and streamline group in the turbulent section. It can be seen that streamlines are in a different plane as they enter the turbulent region, compared to the plane they were in when they were approaching the protrusion.

Figure 62. Turbulent viscosity at $Z = 30$ cm, $t = 0.3$ s: (a) DDES, (b) DES.
As shown in figure 64, prior to the body, flow is turbulent near the lower plate, focusing towards the lower edge of the elliptic frustum. This justifies that shock interaction occurs with turbulent boundary layer. Cross sections of the flow involving the antenna, has a slight indication of turbulent flow. Turbulence intensifies near the intersection between the top of the antenna (i.e., disk) and the antenna stand (elliptic frustum). As one moves further away from the antenna, turbulence intensifies in two regions: lower surface and a region behind the disk-stand intersection. Flow behind the lower section of the stand is more turbulent exactly after the stand
than other regions on the lower surface. This intensity is more visible further away from the antenna, and it creates a gap between lower surface turbulence and after body turbulence. Maximum turbulence is about one chord (50 cm) behind the antenna, near lower section of the disk-stand cross section.
Figure 64. Turbulent Viscosity for \( t = 0.3 \) s and (a) \( x = -10 \) cm, (b) \( x = -1 \) cm, (c) \( x = 10 \) cm, (d) \( x = 20 \) cm, (e) \( x = 30 \) cm, (f) \( x = 40 \) cm, (g) \( x = 50 \) cm, (h) \( x = 80 \) cm, (i) \( x = 90 \) cm, (j) \( x = 100 \) cm

4.2.2 DDES Antenna with Side Slip

Simulations were conducted for the same flow condition with a DDES turbulent model antenna with a ten-degree side slip (simulation performed in this side slip, since this is an extreme case). Figures 65 to 94 show the results (static pressure, temperature, turbulent viscosity) of these simulations.
Figure 65. Static pressure contours at Z = 10 cm, t = 0.15 s.

Figure 66. Static pressure contours at Z = 10 cm, t = 0.3 s.
Figure 67. Static pressure contours at Z = 20 cm, t = 0.15 s.

Figure 68. Static pressure contours at Z = 20 cm, t = 0.3 s.
Figure 69. Static pressure contours at $Z = 26$ cm, $t = 0.15$ s.

Figure 70. Static pressure contours at $Z = 26$ cm, $t = 0.3$ s.
Figure 71. Static pressure contours at Z = 28 cm, t = 0.15 s.

Figure 72. Static pressure contours at Z = 28 cm, t = 0.3 s.
Figure 73. Static pressure contours at $Z = 30$ cm, $t = 0.15$ s.

Figure 74. Static pressure contours at $Z = 30$ cm, $t = 0.3$ s.
Figure 75. Temperature contours at Z = 10 cm, t = 0.15 s.

Figure 76. Temperature contours at Z = 10 cm, t = 0.3 s.
Figure 77. Temperature contours at Z = 20 cm, t = 0.15 s.

Figure 78. Temperature contours at Z = 20 cm, t = 0.3 s.
Figure 79. Temperature contours at $Z = 26$ cm, $t = 0.15$ s.

Figure 80. Temperature contours at $Z = 26$ cm, $t = 0.3$ s.
Figure 81. Temperature contours at $Z = 28$ cm, $t = 0.15$ s.

Figure 82. Temperature contours at $Z = 28$ cm, $t = 0.3$ s.
Figure 83. Temperature contours at Z = 30 cm, t = 0.15 s.

Figure 84. Temperature contours at Z = 30 cm, t = 0.3 s.
Figure 85. Turbulent viscosity contours at $Z = 10$ cm, $t = 0.15$ s.

Figure 86. Turbulent viscosity contours at $Z = 10$ cm, $t = 0.3$ s.
Figure 87. Turbulent viscosity contours at $Z = 20$ cm, $t = 0.15$ s.

Figure 88. Turbulent viscosity contours at $Z = 20$ cm, $t = 0.3$ s.
Figure 89. Turbulent viscosity contours at $Z = 26$ cm, $t = 0.15$ s.

Figure 90. Turbulent viscosity contours at $Z = 26$ cm, $t = 0.3$ s.
Figure 91. Turbulent viscosity contours at Z = 28 cm, t = 0.15 s.

Figure 92. Turbulent viscosity contours at Z = 28 cm, t = 0.3 s.
Figure 93. Turbulent viscosity contours at \( Z = 30 \) cm, \( t = 0.15 \) s.

Figure 94. Turbulent viscosity contours at \( Z = 30 \) cm, \( t = 0.3 \) s.
In order to investigate shock wave turbulent boundary layer interaction, flows around two protrusions were simulated using two numerical schemes, namely detached eddy simulation (DES) and delayed detached eddy simulation (DDES). These two numerical models were consistent in using a Reynolds average Navier-Stocks (RANS) model for near-wall sections and a large eddy simulation (LES) model for regions where eddies are presented. The RANS formulation with Spalart-Allmaras (SA) turbulent model was used for both DES and DDES. Flow properties in the two configurations were presented, and the two numerical schemes were compared.

The first protrusion was a bump. Simulations took place with DES-SA turbulence model, and results were compared with experimental and numerical data. Comparing static pressure on the lower surface (bump location) for numerical and experimental results validated the DES-SA turbulent model. A comparison of the DES-SA and experimental data to the $k-\omega$ and experimental data, showed that the DES-SA model managed to capture flow conditions more accurately in the near-wall regions. For other locations in the computational domain, the results were similar to each other. Flow properties in the middle of the 3D computational domain were similar to the 2D results, which was similar to the middle section of wind-tunnel results.

Another observation in this simulation is that regions closer to side walls have different flow characteristics than those closer to middle section of the domain. For a distance sufficiently far from the side walls, as mentioned, flow characteristics are the same as a 2D simulation. The effect in which flow properties closer to the side walls are different than the center section of the
computational domain is called the wall effect. Wall effect increases the turbulent region of the flow (near lower surface) and also weakens the shock wave.

Wall effect could be a sub category of the three dimensionality effect, or the 3D effect. The 3D effect is more visible at the corner of the computational domain which produces circulation after the bump. These phenomena create a different flow pattern for the outflow. Inflow is a unified mass entering the domain, with visible boundary layer around it. All sections of the inflow are symmetrical with respect to the center point of the inlet duct. After flow passes over the bump on the lower surface, flow pattern between the lower and upper section of the computational domain changes in a way that the lower edge of free stream mass flow at outflow becomes closer to a circular section, as the flow closer to the upper wall is still similar to the inflow’s pattern. DES model doesn’t allow the usage of a half-model. Half-model is when the flow is simulated only for half of the domain and a boundary is set to be symmetrical. If a RANS model was used, only half the width of the wind tunnel was required. This would noticeably reduce the computational time.

Inflow for the bump model is a fully developed flow in a 50 cm channel. The flow entering that system was uniform. After flow passes through the channel, it creates a 7 mm boundary layer in which the flow velocity near the wall regions changes from zero (assuming the immediate layer of the flow after the wall is stationary) till around 7 mm from the wall where flow has the maximum velocity, called the free stream velocity. Boundary layer on the lower surface varies by the distance it has from the side wall. That is natural, since the side walls have also a boundary layer, and affects the lower surface boundary layer.

Final observation in this model is the flow circulation in two sections: circulation near the side walls and circulation near the lower surface. First circulation is due to 3D effect and moves
the flow path from the surface to the side wall. Plotting static pressure on the lower surface at 
y=1 cm shows variation on the rate of static pressure increasing after the shock location due to 
this circulation. Second circulation occurs due to down wash. Since the flow is sliding down the 
bump, the flow mass is forced inward towards the duct (beside those captured in the corner flow 
circulation) and masses of flow from locations closer to the walls before the bump are directed to 
the middle section of the bump.

For future work, this design and 3D effect could be used to solve a common problem for 
high speed engine inlets. A common problem in these engines is the shock wave occurrence 
before the combustion chamber. The 3D effect manages to weaken the shock wave, or to 
terminate shock waves inside the inlet.

The second protrusion was an aircraft UHF antenna design. Flow was simulated using 
DES-SA and DDES-SA numerical models. The antenna is a disk with round edges placed on an 
elliptical frustum. Results from the two numerical models were presented and compared. Due to 
its nature, the DDES model managed to preserve the boundary layer around the object and 
simulate the shock wave in a location most expected by theory. Both DES and DDES results 
were reasonable for inward flow and flow properties after the shock location.

The intersections regions, disk-stand and stand-lower surface, are the main disruption 
sources that create high turbulent sections and slightly abates turbulence for flow after the near 
stand-lower surface area. Closer observation shows masses of flow are driven down and closer 
towards the antenna, thus creating the turbulent regions. Streamlines before the stand are either 
sent to the lower surface, or semi-rotating the stand and passing along the disk-stand connection. 
Flow on top of the antenna, up to 15 cm, is driven down towards antenna’s disk. The antenna 
affects flow up to 25 cm on top of the antenna (close to two times the antenna’s height) but
doesn’t affect the flow stream beyond that. Since free stream flow velocity is high, the turbulent region doesn’t spread out much and it shrinks after 50 cm behind the antenna.

The flow simulation was continued using a DDES model with the same flow conditions at a 10-degree side slip. This side slip is rather large for this object’s nature, but simulating an extreme case was the main purpose of this assumption.

Results between DES and DDES model were compared based on analytical theory. The elliptical frustum had a small aspect ratio, and flow around was expected to create a shock wave towards the back of the object. The exact location of the shock wave and flow properties could not be validated by these simulations, unless a reliable experimental result could be presented.
REFERENCES


