Theses and Dissertations

2005-07-27

Incorporating Computational Fluid Dynamics Into The Preliminary Design Cycle

Jonathan Knighton Shelley
Brigham Young University - Provo

Follow this and additional works at: <https://scholarsarchive.byu.edu/etd>

Part of the [Mechanical Engineering Commons](#)

BYU ScholarsArchive Citation

Shelley, Jonathan Knighton, "Incorporating Computational Fluid Dynamics Into The Preliminary Design Cycle" (2005). *Theses and Dissertations*. 631.

<https://scholarsarchive.byu.edu/etd/631>

This Thesis is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.

INCORPORATING COMPUTATIONAL FLUID DYNAMICS INTO THE
PRELIMINARY DESIGN CYCLE

By

Jonathan Knighton Shelley

A thesis submitted to the faculty of

Brigham Young University

in partial fulfillment of the requirements for the degree of

Master of Science

Department of Mechanical Engineering

Brigham Young University

August 2005

Copyright © 2005 Jonathan Knighton Shelley

All Rights Reserved

BRIGHAM YOUNG UNIVERSITY

GRADUATE COMMITTEE APPROVAL

of a thesis submitted by

Jonathan Knighton Shelley

This thesis has been read by each member of the following graduate committee and by majority vote has been found satisfactory.

Date

C. Greg Jensen, Chair

Date

Kenneth W. Chase

Date

Spencer P. Magleby

BRIGHAM YOUNG UNIVERSITY

As chair of the candidate's graduate committee, I have read the thesis of Jonathan Knighton Shelley in its final form and have found that (1) its format, citations, and bibliographical style are consistent and acceptable and fulfill university and department style requirements; (2) its illustrative material including figures, tables, and charts are in place; and (3) the final manuscript is satisfactory to the graduate committee and is ready for submission to the university library.

Date

C. Greg Jensen
Chair, Graduate Committee

Accepted for the Department

Matthew R. Jones
Graduate Coordinator

Accepted for the College

Alan R. Parkinson
Dean, Ira A. Fulton College of Engineering and
Technology

ABSTRACT

INCORPORATING COMPUTATIONAL FLUID DYNAMICS INTO THE PRELIMINARY DESIGN CYCLE

Jonathan Knighton Shelley

Department of Mechanical Engineering

Master of Science

Industry is constantly looking for ways to bring new or derivative products to market in the shortest amount of time for the least amount of money. To accomplish this, Industry has adopted Computer Aided Engineering (CAE) tools that perform structural, flow, manufacturing, and cost analysis. The way in which a company utilizes these CAE tools can determine the success of these new products.

One of these tools that Industry often struggles with in the preliminary design of a product is Computational Fluid Dynamics (CFD). Some of the challenges presented by CFD are the time it takes to create a CAD model, generate a valid grid for analysis, obtain a solution, post-process results, and review the result. The objective of this thesis is to develop an approach that will reduce the time it takes to go from a concept to a

solution ready for review. This approach, addresses how to first, build CAD model for use in downstream applications, second, automate the grid generation process, and third, automate the post process and documentation of the results.

Using this approach a conceptual study of a two stream mixing problem was performed. The approach showed that the creation of the first model took about twenty percent longer than the standard practice used in industry today. However, once the first model was completed, different concepts could be added to the CAD model and be ready for analysis in less than half the time when compared to standard practices. This time savings can then be used to explore more concepts. After each model was analyzed, it was post-processed using an automated script. With the post-processed results the Design Review Tool (DRT) was developed to automate the documentation of the results. Using the DRT each post-processed case was organized into a web page and saved for review in less than five seconds.

This approach will enable the aerospace, automotive, and other industries to use CFD to more effectively explore the design space in the development of new and derivative products. This research demonstrates the process to reduce the time required to go from CAD-to-Grid, postprocess the results, and create the documentation needed to develop new products.

ACKNOWLEDGMENTS

I would like to first thank my wife and children for their love and support. Without their support this never would have happened. Second, I want to thank Dr. C. Greg Jensen, Dr. Stephen Kramer, and the BYU faculty for their encouragement, friendship, help, and support throughout my graduate studies. Lastly I want to thank Pratt and Whitney for supporting this research.

TABLE OF CONTENTS

CHAPTER 1: INTRODUCTION.....	1
1.1 MOTIVATION.....	2
1.2 THESIS OBJECTIVE	3
1.3 DELIMITATION OF THE PROBLEM	4
CHAPTER 2: LITERATURE REVIEW:	5
2.1 PRELIMINARY DESIGN USING CFD.....	5
2.1.1 <i>Parametric CAD Modeling for CFD</i>	6
2.2 LINKING COMMERCIAL CAD WITH CAE TOOLS.....	6
2.3 GRID GENERATION.....	8
2.4 RESULTS VISUALIZATION.....	8
CHAPTER 3: METHOD	11
3.1 CAD MODELING FOR CFD IN PRELIMINARY DESIGN	11
3.1.1 <i>Construct a Master Cross Sectional Sketch (MCSS)</i>	12
3.1.2 <i>Model the Component Using the MCSS References</i>	13
3.1.3 <i>Assembly of the Various Components</i>	14
3.2 GEOMETRY PROCESSOR	15
3.3 GRID GENERATION.....	15
3.4 CFD ANALYSIS	16
3.4.1 <i>Defining the Boundary Conditions</i>	16
3.4.2 <i>Preprocessing</i>	16
3.4.3 <i>Analyzing the Model</i>	17
3.4.4 <i>Post Process Results</i>	17
3.5 DESIGN REVIEW TOOL	17
CHAPTER 4: DEVELOPMENT	19
4.1 CAD MODEL	20
4.1.1 <i>Constructing the MCSS</i>	20
4.1.2 <i>Component Modeling</i>	23
4.1.3 <i>Assembly Modeling</i>	27
4.2 GEOMETRY PROCESSOR	31
4.3 GRID GENERATION.....	34
4.4 CFD ANALYSIS	36
4.4.1 <i>Preprocessing</i>	36
4.4.2 <i>Boundary Conditions</i>	37
4.4.3 <i>Analyzing the Model</i>	38
4.4.4 <i>Postprocess the Results</i>	38
4.5 DESIGN REVIEW TOOL	39

CHAPTER 5: DISCUSSION OF RESULTS	49
5.1 RESULTS: AUTOMATED GRID GENERATION OF PARAMETRIC MODELS	49
5.1.1 Results: MCSS.....	50
5.1.2 Results: Component Modeling.....	50
5.1.3 Results: Assembly Model	50
5.1.4 Grid Generation.....	53
5.2 RESULTS: DESIGN REVIEW TOOL	55
5.2.1 Does the DRT present the results effectively?	56
5.2.2 Are the results presented in a timely manner?.....	56
5.2.3 Are results of each case setup to be archived for later use?	56
CHAPTER 6: CONCLUSIONS	59
6.1 FUTURE WORK.....	61
REFERENCES:	63
APPENDIX.....	67
APPENDIX A: SETUP INFORMATION AND FLOW CALCULATIONS	69
APPENDIX B: DRT SCRIPTS	71
APPENDIX C: DRT SCREEN GRABS FOR CONCEPT 001.....	81

LIST OF FIGURES

Figure 3-1	Elements for incorporating CFD into preliminary design.....	11
Figure 3-2	Defining a master cross sectional sketch	12
Figure 3-3	Topology changes to a surface.....	14
Figure 4-1	Rolls-Royce Trent combustor (Cervenka 2000)	19
Figure 4-2	Master Cross Section Sketch.....	21
Figure 4-3	Entities referenced from the MCSS for the front-end concepts	21
Figure 4-4	Entities referenced from the MCSS for the ID and OD liner concepts.....	22
Figure 4-5	Front-end concepts	23
Figure 4-6	Liner orifice concepts.....	24
Figure 4-7	Before and after the model was trimmed	24
Figure 4-8	Surfaces tagged for mesh generation.	26
Figure 4-9	Family, faces, and curves tagged for front-end concept #3	27
Figure 4-10	Initial air solid	28
Figure 4-11	Sheet bodies used to close the air solid.....	28
Figure 4-12	Air solid model.....	29
Figure 4-13	Assembly issues when a component is toggle off and another one is toggled on.....	30
Figure 4-14	Air solid with 3 periodic faces	31
Figure 4-15	With and without running the geometry processor	34
Figure 4-16	Flow velocities into the plenums.....	37
Figure 4-17	A screen grab of the preliminary CFD study page.....	41
Figure 4-18	DRT: Case template	42
Figure 4-19	DRT: Model section.....	43
Figure 4-20	DRT: Grid section.....	43
Figure 4-21	DRT: Flow conditions section	44
Figure 4-22	DRT: Pressure map section.....	44
Figure 4-23	DRT: Flow results section.....	45
Figure 4-24	DRT: Temperature results page	46
Figure 4-25	DRT: Temperature slice information image	46
Figure 4-26	DRT: Objective section.....	47
Figure 5-1	Base assembly with front-end concept #1 and liner hole concept #3	51
Figure 5-2	Case 002 air solid assembly with front-end concept #2 and liner hole concept #1	51
Figure 5-3	Case 006 air solid assembly with front-end concept #3 and liner hole concept #2	52
Figure 5-4	ICEM screen capture of the input file that was imported from Unigraphics	53
Figure 5-5	The finished grid that was generated for case004.....	54

CODE FRAGMENTS

Code Frag 4-1	Code used to open the CAD model, read in the expression file and update the model	32
Code Frag 4-2	Find the specified family edges and add points and tag them	33
Code Frag 4-3	Script created to automate the grid generation process	35
Code Frag 4-4	Code to import geometry from Unigraphics and add density volumes	35
Code Frag 4-5	Ensign commands to read in the model and results.....	38
Code Frag 4-6	Input needed to define the location of the files for the DRT	39
Code Frag 4-7	Model class module for the DRT	40

INTRODUCTION

Today's highly competitive market place demands high quality, low-cost products, developed in a timely, rapid fashion. According to Ulrich and Eppinger (2001), effective product development and release requires: planning, concept development, system-level design, detailed design, testing and refinement, and production ramp-up. Conceptual and preliminary design steps define product discriminators and provide improved quality and lower cost. Automated best-in-class conceptual and preliminary design tools facilitate these steps.

Historically companies have been unable to explore all possible concepts and combinations of concepts due to limited resources and competitive pressures. Concepts have and are being carried forward to the detailed design phase based on partial information, partial exploration of the design space, and insufficient development. For most companies still challenged by limited resources and pressures from their competition, their detailed design phase is littered with technical challenges and roadblocks caused by premature termination of the conceptual and preliminary design phases.

Ideally, all possible concepts would be explored before selecting the one that best meets requirements and is deemed ready for detailed design and production. Tools like computer-aided design (CAD), finite element analysis (FEA), and computational fluid dynamics (CFD), provide simulations that are used to numerically compare concepts and help the engineer understand the problems and challenges of particular concepts.

However, Haimes (2003) states, and personal experience in industry confirms, that getting the first CAD model(s) ready for analysis tends to take weeks to months. CAD models are being integrated with CAE tools as faster and more capable computers become available (Alonso et al. 2003). Limiting factors for these integrated design tools becomes user knowledge, modeling techniques, software capabilities, and computer hardware resources.

1.1 Motivation

Today, in industry a fluids engineer works with the designer to create an air solid for a given concept. Typically a cross section is given to the designer to define the 2-Dimensional shape. The cross section is commonly unparameterized. Using this unparameterized cross section as a reference the designer creates the air solid. When the air solid is completed, the designer passes the model to the grid specialist. The grid specialist looks over the model and generates the first grid. If there are problems, i.e. leaks, misalignment of surfaces, slivers, etc, the designer is asked correct the model. The designer then changes the model and gives it back to the grid specialist. The grid specialist then creates the final grid. If the fluids engineer wants to run variants of the model or explore different topology concepts, he works with the designer to create the new air solids. This can take anywhere from a few hours to a few weeks. The air solids are then handed to the grid specialist and depending on the changes, it takes between a few hours to a week to generate each new grid. This current process is time consuming, costly, and involves multiple people.

To better utilize the array of modern CAD/CAE tools and the company's computing resources a methodology is needed, which reduces the time required to go

from CAD/to/Mesh. This timesaving could then be used to explore more concepts before having to select a particular design to carry forward to the detailed design phase.

Another challenge to using CFD in conceptual and preliminary design is the amount of data that must be processed and presented. Currently, the standard practice in industry is to post process the CFD solution and generate images for a set of parameters. The analyst then proceeds to make a presentation. The presentation consists of a discussion of the model, a review of the boundary conditions, and pressure field, and then a review of the results such as temperature, velocities, etc. If performed manually, the time it takes to document each case can take hours. When iterations of the model are performed, the analyst commonly takes the previous presentation as a template and updates it. This process of updating the presentation results in delaying the review of the results until a formalized presentation can be made. A method is needed which will allow data to be processed in a consistent manner and formatted for quick review.

1.2 Thesis Objective

The focus of this thesis will be to develop a method that integrates the parametric capabilities of CAD systems with CFD analysis tools in order to solve some of the challenges discussed above. Solving these challenges will enable the conceptual and preliminary design teams to evaluate significantly more concepts in a timely fashion. Once the method is proven, the author believes that the geometric complexity, quantity of engineering knowledge embedded and the frequency of tool reuse will naturally increase.

To illustrate the methodology developed in this thesis a two stream mixing problem will be analyzed. The following specific questions will be addressed:

1. Can valid CFD grids be made in an automated way from parametric models that undergo predictable topology changes?
2. Can valid CFD grids be made in an automated way when one concept is replaced with a different concept in an assembly?
3. How can CFD data be evaluated more efficiently for design reviews?
4. How can CFD data be effectively documented for later use?

The CAD models will be created using Unigraphics (UG). ICEMCFD commonly referred to as ICEM, will be used to create the grids required for the CFD analysis package. The CFD solver will be Pratt and Whitney's in-house code All-Star. Ensign visualization software will be used process the CFD data. To link the various tools together, as well as format the data for future use, the scripting language, Python will be used. It is anticipated that by answering these questions, design teams in industry will be able to explore more concepts using CFD before a concept is selected and moved into the detailed phased.

1.3 Delimitation of the Problem

The author believes that the method presented in this work is general enough to be applied to the various commercial CAD and CAE tools. However, for this thesis only the tools mentioned above will be considered. Every attempt will be made to specify loads, boundary conditions, constraints, etc. so that the CFD model will conform to industry practices.

LITERATURE REVIEW:

Research work and topics that form the foundation on which this research will build are: preliminary design using CFD, parametric modeling for CFD, linking commercial CAD to CAE tools, grid generation, and presentation of results. In this prospectus a brief summary is given as to current work that supports the direction of this thesis. Where possible, I have included quotes that support and encourage the direction being pursued.

2.1 Preliminary Design using CFD

CFD modeling of complex geometry in the aerospace industry is commonly used in the design process (Crocker et al.1998). Incorporating CFD modeling of assemblies into the preliminary design process presents challenges due the time required to generate the first grid and subsequent grids. Grid generation "is the most labor intensive and time consuming part of computation aerosciences" (Samareh 1999). Haimes et al. (2003) states that the time to go from CAD to generating the first CFD case can take weeks to months. The bulk of the time and labor is spent simplifying the geometry, exporting the model to the CAE tool, repairing the geometry if needed, and tagging the model for gridding. For these reasons Smith (1996) states that research is needed to eliminate grid generation time so that more time can be used to incorporate analysis more fully into the concept generation stage.

2.1.1 Parametric CAD Modeling for CFD

Parametric CAD systems use a “feature tree” approach to construct the solid model. Each feature created is updated in the order that it was created. Features consist of sketches, lines, curves, points, surfaces, solids, etc. Using these features in a tree structure enable the solid model to update easily and have good reusability characteristics (Hoffmann 2001).

CAD is the enabling technology for the creation of solid models, design changes, and down stream analyses (Lee 1999). Ault (1999) states that the development of the CAD model is critical to maintain design intent. Today most engineering analyses use parametric solid model(s) as the foundation for the analysis models (Srinivassen et al. 2001). If a model is built parametrically, design changes can be easily performed (Anderl 1995). The parametric model can be modified by either manually modifying the expressions or by reading in an expression file (Rohm et al. 2000).

2.2 Linking Commercial CAD with CAE Tools

Integration of CAD tools to downstream CAE applications becomes an enabler to improving the product development process (Armstrong 2002). Today, there are two approaches to transferring the data from the CAD system to the CAE tool. They are by neutral file format (i.e. STEP and IGES) or direct translators. Krause et al. (1997) discusses an approach to working with neutral formats and how to repair errors caused during the translation in the geometry. If the model has been translated using a neutral file format, the user is required to assign entity parameters to each surface and curve. This method is fine if the model will only be gridded once.

Direct translators provide a one directional link from the CAD model to the CAE tool. This integration of the CAD and CAE tools for passing complex geometry from the CAD tool to different CAE tools in a reliable and timely fashion has only been available for the past few years (Dawes et al. 2000, Tangirala et al. 2000). Today's direct interfaces allow the "generation of complex-geometry grids without requiring boundary conditions and mesh sizes to be respecified" (Malecki et al. 2001). They do this by storing the necessary information needed for the grid generation phase in object attributes. Subrashekar et al. (1995) said attributes are "ideal candidates to store information relevant to a particular phase in the life-cycle of a product. For EXAMPLE: design, analysis, assembly, process planning, etc."

However, one of the current limitations to using a direct translator in an automated way is that parametric changes to the geometry can result in topological changes (Hardee et al. 1999). These topological changes can result in new objects that have not been assigned. This will result in an invalid grid when the model is translated and gridded. To address some of these issues, King (2004) developed a tool that would look at feature arrays and User Defined Features (UDF's) and apply the attributes from the parent feature in the array to all of the instances. While this addresses one set of topology problems, it does not address problems such as a holes on a periodic plane or surface that are broken in two resulting in four edges for the hole instead of two, or a hole that lies on a subdivided face resulting in eight edges instead of four. This thesis will look at addressing the topology changes mentioned above that are not part of an array.

2.3 Grid Generation

Crocker (1998) states that “generating a quality grid in a reasonable amount of time” is a significant challenge. Usually, the CAD model has already been designed. For the model to be used in CFD it generally needs simplification. This requires suppressing holes, rounds, chamfers, etc. These simplifications are often performed in the CAD package due to the poor modeling capabilities of the gridding tool. For complex geometries the common gridding technique used is an unstructured grid. Unstructured grids enable the gridding process to become more automated because “they are capable of resolving irregularly shaped domains more easily and with greater efficiency” (Newman et al. 1999). Newman further states that unstructured grids can be “adapted and locally enriched where needed without affecting other regions of the mesh.” Although unstructured grids provide the flexibility to grid all of these irregular shapes, some geometry contains problem spots that require hand fixing. This need for hand fixing prevents the process to be fully automated.

2.4 Results Visualization

As computer tools evolve, they enable the designers to generate more and more data. Hogge (2002) states, that after a design study has been performed it can be challenging to sort thru all the data. Most of these visualization tools have two or three ways for the engineer to get at the data. They are the GUI, macros, and API's (Hogge, 2002). For small concept studies or preliminary design projects API are poorly suited due to the development time required versus an interactive approach and the developer must know the API of the particular system being used (Rohm, 2001). For these reasons, this

research looks to make use of macros to process the visualization data due to the ease in creating them for small conceptual studies.

METHOD

The following paragraph describes the union of various commercial and noncommercial CAx tools. Currently, the CFD process is manual, error-prone, and too time intensive for use in preliminary design of complex models. While no significant research and development is added to the commercial and noncommercial tools themselves, this work will focus on the development of interfacing the different tools. The following approach will describe in detail the steps that will be used to handle some topology changes as well as replacing components in the assembly model which goes beyond the common parametric changes to an air-solid used in industry today.

Figure 3-1 Elements for incorporating CFD into preliminary design

3.1 CAD Modeling for CFD in Preliminary Design

The CAD modeling strategy can be summarized in three steps;

- a. Construct a Master Cross Sectional Sketch (MCSS)
- b. Model the component using the MCSS references
- c. Assembly of the various components

If these steps are followed in the consistent and standard way as disclosed below the result will be a robust and reusable model.

3.1.1 Construct a Master Cross Sectional Sketch (MCSS)

The purpose of the MCSS is to define key references and features for an assembly. Before a MCSS can be constructed, these key references and features (i.e. Datum planes and axes, Assembly features, Component locations, etc) need to be defined. Below is an example of using datums to define the location of square box and cylinder in an assembly using a MCSS.

Figure 3-2 Defining a master cross sectional sketch

In the figure above there are four datum planes, three datum axes and one sketch. The datum plane with the sketch of the square and the circle would be an assembly plane or interface where two components would be positioned. By using common datums in the MCSS to locate the different components reduces the time required to create the assembly. Also, changes in the MCSS will propagate through the entire assembly in a consistent manner.

Once the master sketches are created, the designer(s) can work off the same sketches to make certain that the interface points and locations are consistent. This guarantees that the components will match up properly when it comes to populating the assembly and eliminate unintentional gaps.

3.1.2 Model the Component Using the MCSS References

In order for the components to take advantage of the information stored in the MCSS Inter-part geometry and Inter-part dimensioning will be used. Inter-part geometry linking provides a way for geometry defined in the MCSS to be referenced by the different components. The linked geometry enables the various components to be adjusted to changes that occurred in the MCSS. If used properly, inter-part geometry linking helps to ensure that the overall assembly will maintain design intent during design iterations.

Inter-part dimensioning allows information from the MCSS to flow to the individual components. This is done by creating assembly level parameters in the MCSS and linking them to the individual components. For example, if the overall length of the square and the circle defined in the example above were to be ten inches long, parameters such as

$$\text{Total_length} = 10$$

$$\text{Box_length} = \text{Total_length}/3$$

$$\text{Cylinder_length} = \text{Total_length} - \text{Box_length}$$

can be defined. Parameters such as these will be defined in the MCSS and referenced by the components to define their critical dimensions.

Once the component is modeled, attributes needed for mesh generation will be assigned to the different geometric entities. Defining the element size for each face and

curve often requires a few iterations before the different features are properly resolved. When the analyst is satisfied with the grid, this information can then be used for subsequent designs. However, when the topology is changed the analyst needs to recognize the topological changes, tag any new geometric entities, and ensure the new entities will be properly resolved. For example, the top face on the left block (below) needs nine entities to properly resolve its shape, while the right block requires an additional edge to resolve its top face.

Figure 3-3 Topology changes to a surface

3.1.3 Assembly of the Various Components

Once the components are modeled, they will be added to the assembly. If the component modeling step was performed correctly, the various components should require little to no information in order to be assembled correctly. At the assembly level, interface surfaces may be needed to close any gaps between the various components. These interface surfaces may be tied to expressions in order to toggle them on or off , based on a particular configuration.

Assembly expressions will be used to toggle the parametric components on/off as well as any interface surfaces. This will allow an input file to modify the assembly in order to represent the different concepts.

3.2 Geometry Processor

The geometry process step is used to sort through the model looking for any entities, such as edges, that are not currently tagged and assign attributes as needed. For predetermined topological changes, such as the one in Figure 3-3, the geometry processor will need to be able to recognize the new topology and tag the appropriate entities. The geometric process will need to do the following:

1. Collect the attribute names that may contain topology changes
2. Modify the expressions and update the CAD model
3. Search the entities for specified attributes names
4. Check entities to see if they are tagged
5. Assign the proper attributes to any new topology
6. Save the model

3.3 Grid Generation

ICEM CFD will be used to create the grid. The first grid will be created interactively to determine the steps required. Each step will be recorded so that the steps can be incorporated into a script. This script will then be used to perform the following steps necessary for gridding.

1. Adding density volumes
2. Generating the grid
3. Smoothing the grid

4. Saved the grid for review.

The review step is necessary to verify and fix the grid, if needed, before it is preprocessed.

3.4 CFD Analysis

3.4.1 Defining the Boundary Conditions

Boundary conditions are the inputs required to solve the differential equations used in CFD analysis. Some common boundary condition types are inlet speeds, walls, periodic faces, symmetry planes, etc. Initial boundary conditions must be set before the model is solved. These boundary conditions can then be modified to match the desired conditions. Once these conditions are set they usually require minor changes, if any, for the subsequent derivative models.

When performing a CFD analysis, there are three steps that are required in order to obtain the results. The first step is preprocessing the grid

3.4.2 Preprocessing

For this thesis, standard practices were used for the preprocessing. These standard practices include checking the grid for errors, assigning the boundary conditions to the nodes, and breaking up the model to be run on multiple computers. When the grid is ready, the next step is analyzing the model

3.4.3 Analyzing the Model

As stated in the introduction, the focus of this thesis is on the model preparation process and not the analysis results. Since no experimental data is available to validate the results, CFD results will be assumed to be valid and provide evidence that the model preparation is adequate. When the CFD solver has converged to a solution the final step is to post process the results.

3.4.4 Post Process Results

Enight visualization software was used to perform the post processing for each converged solution. To automate this step a command file (macro) must be created for the first case. In general the command file will need to

1. Open the analysis model
2. Load the results file
3. Calculate desired variables for study
4. Generate images of the model
5. Output the data
6. Exit the program

Once the macro file is setup, it will perform all of the subsequent postprocessing in a consistent way.

3.5 Design Review Tool

The Design Review Tool (DRT) is created to organize the results gathered from the post processing step in a timely fashion. It also formats the results for later use. To do this, a standard file structure is required to allow the DRT to organize the data into a set

of web pages that can be easily reviewed by other design team members. The web pages contain

1. Images of the concepts
2. Tables containing flow conditions and results
3. Graphs displaying the results

By presenting this data effectively in an automated fashion, the time required to prepare a design review should be significantly reduced. It also enables the design team to look at the results as soon as they are available as well as storing them for later use.

DEVELOPMENT

The method presented in chapter 3, as shown in Figure 3-1, was applied to a two stream mixing problem. One example from industry of a two stream mixing problem is a combustor in a gas turbine engine shown in Figure 4-1. As shown the fuel spray nozzle injects the first stream or hot stream. The holes in the liners allow the injection of the second stream or cold stream.

Figure 4-1 Rolls-Royce Trent combustor (Cervenka 2000)

For this study, a simplified representation of a combustor was used. It consists of three different components. These components are a front-end, which introduces the hot

stream and the outer and inner liners, which introduces the cold stream. Different concepts and configurations were explored for each component to determine which concept provided the most uniform temperature at the exit plane.

4.1 CAD Model

The foundation for the two-stream mixing problem is the CAD model. The following sections present the steps used for the creation of the components and assembly.

4.1.1 Constructing the MCSS

For the two stream mixing problem, a MCSS was created. In the MCSS the envelope of the assembly, datum references, and assembly interfaces are specified. First, to define the origin, three datum planes were created. Second, to define the centerline of the two-stream mixing problem a datum axis was created. Third, the assembly envelope, containing the mixing chamber and the OD and ID plenums was laid out. Fourth, the injection location for the second stream of air was added. The MCSS for the simplified two stream mixing problem can be seen in Figure 4-2.

Figure 4-2 Master Cross Section Sketch for the two stream mixing problem

From the MCSS, the different components reference the necessary entities. The datums, lines and point that are referenced by the front-end concepts are shown in Figure 4-3.

Figure 4-3 Entities referenced from the MCSS for the front-end concepts

The outer and inner lips indicate where the front-end mates up to the liners, while the Injection point is used by the different front-end concepts to locate their injection streams.

The liners reference the datums and entities as shown in Figure 4-4. These objects are used to define the inner and outer profiles, panel thickness, and the injection location for the cold stream

Figure 4-4 Entities referenced from the MCSS for the ID and OD liner concepts

The dashed line defines the distance d downstream of the inlet, where the second stream is introduced. The shorter lines, on the top and bottom, represent the length l_1 of the outer and inner plenums. They also define the liner thickness t . The mixing chamber wall lines define the length l_2 of the mixing chamber. The points on the longer lines define where the front-end components will mate up with the liners. After the MCSS is defined, the next step is to model the components.

4.1.2 Component Modeling

Each component was modeled by first adding the MCSS. After that by a technique called waving, the datums, curves, and points in the MCSS were added to the component. In Unigraphics, the concept of waving geometry allows the referencing of objects from one part to another. Standard dimensioning practices generally forbid chaining dimensions in this manner. But, by referencing these objects, any changes to the MCSS will propagate through all of the components.

For the front-end, three different concepts were selected. The first concept has an array of four rectangular openings. The second has a cylinder with a flow separator in the middle. The third has a set of four cylindrical tubes. These three different front-end concepts are displayed in Figure 4-5.

Figure 4-5 Front-end concepts

For the different liner concepts, three orifice shapes were selected. The shapes of the orifices used to inject the second stream are circular, elliptical, and triangular. These orifice concepts are shown in Figure 4-6

Figure 4-6 Liner orifice concepts

When constructing these concepts, certain modeling techniques were applied to ensure a valid model. First, the liner was revolved beyond the actual sector width. Second, the holes were added with the centroid of the area matching up with the second stream injection location. Each orifice was created separately to ensure the face IDs would not change. Third, all faces and curves needed for the mesh generation were tagged. Fourth, the liner was trimmed to the appropriate sector size. The reason for this is sometimes the holes will be on the periodic boundaries, which require one complete hole and two half holes. Other times the holes might not be on the periodic boundaries leaving only two complete holes. This can be seen in Figure 4-7.

Figure 4-7 Before and after the model was trimmed

With the concepts modeled, the next step is to match the flow area for each component concept to ensure a fair comparison. To calculate the flow area (AC_d) needed for the desired pressure drop, the following equation was used.

$$AC_d = \frac{\dot{m}}{\sqrt{\frac{2\Delta p}{\rho}}} \quad 0-1$$

A is the geometric area of the orifice, C_d is the discharge coefficient, $\dot{m}$ is the mass flow rate, ρ is the density of the fluid, and Δp is the difference between the total pressure upstream and down stream static pressure. The C_d parameter was used to properly size the geometric area (A) for each component. The parameters shown in List 0-1 were used to adjust the geometric area for the elliptical orifice to match the correct flow area.

```
c_inlet_Cd=.75
c_flow_area=tm_assy::a_id_flow_area
c_geom_area=c_flow_area/c_inlet_Cd
```

List 0-1 Parameters used to calculate the necessary geometric area

In the pseudo code C_d is C_d . The user sets c_inlet_Cd . Next the expression c_flow_area is passed from the assembly to the components, and then c_geom_area is found by dividing the AC_d by C_d . After the geometric area was determined, the geometric area is updated.

For round holes with sharp edges and a length over diameter (L/d) around 0.2 the C_d is approximately 0.7. For the oval and triangular holes the C_d is unknown and must be determined. To do this a C_d was guessed for the oval and triangle holes. A guess of 0.75 and 0.6 were selected respectively. To find the actual C_d for these holes, CFD simulations

were performed. The C_d for the holes was calculated by using Equation 4-2 at the measured pressure drop from the CFD simulations. The values calculated for C_d for the different concepts are shown in Appendix A.

After front-end and liner concepts were modeled, they were assigned object attributes to resolve the geometry for the mesh generation step. The required faces, curves, and points were tagged using ICEM's UG Mesher interface tool. For example, the faces tagged for front-end concept #3 are shown in Figure 4-8.

Figure 4-8 Surfaces tagged for mesh generation.

The gray face on the right was tagged as the inlet family. The rest of the lighter faces were tagged and assigned to their respective families. The edges of the selected faces were also tagged. A screen grab from ICEM shows the families, faces, and curves tagged for the mesh generation step in Figure 4-9.

Figure 4-9 Family, faces, and curves that were tagged for front-end concept #3

The other front-end concepts were tagged in a similar manner. For the liners, just the surfaces were tagged and assigned families, since they will be checked with the geometry processor.

4.1.3 Assembly Modeling

Adding the MCSS first, and then the components, the assembly model was constructed. To define which components are present for a given configuration, “controlled by expression” parameters were used to toggle them on and off. Next, an air solid was created using entities from the MCSS and revolving them. The air solid without the front-end or liners subtracted can be seen in Figure 4-10.

Figure 4-10 Initial air solid

As seen in the figure above, there is no solid portion for the different front-end concepts to be subtracted from, due to the different topologies for each front-end. The different front-end concepts have added to them two sheet bodies to provide periodic and inlet(s) surfaces in order to close off the air solid. These added sheet bodies are shown in Figure 4-11

Figure 4-11 Sheet bodies used to close the air solid

Next, the OD and ID liners were subtracted from the air solid. Figure 4-12 shows the resulting air solid.

Figure 4-12 Air solid model

When the components were added to the assembly and removed using “controlled by expression”, a few problems arose. First, a linked body was used to subtract the component from the air solid. When one component is toggled off and another on, the linked body of the inactive component remained, causing two of the same component to be subtracted from the air-solid. For example, when the triangular orifice was unsuppressed, the oval orifice remained. This is shown in Figure 4-13.

Figure 4-13 Assembly issues when a component is toggle off and another one is toggled on

To resolve this issue, a feature set was created for each linked body and subtraction feature. Then, an expression was created, that would suppress the feature set when the component was suppressed. When this was done the assembly model updated correctly for all configurations.

With the air solid completed, the assembly was changed to a state which contained all of the potential periodic faces. This was done, by moving the injection holes on the ID and OD liners off of the periodic faces. The three periodic faces which resulted, are shown in Figure 4-14. These faces are the dark faces on the side of the OD and ID plenums and the lighter face on the mixing chamber.

Figure 4-14 Air solid with 3 periodic faces

In this configuration, the three faces were tagged as periodic faces. This was done to ensure that any new configuration of this model would have the periodic faces tagged. The model was then saved in this state as the master model. The model at this point was ready for the geometric processor.

4.2 Geometry Processor

The geometry processor is used to prepare the CAD model to be gridded. The geometric processor reads in the input file, opens up the CAD model, imports the expression file, updates the model, and check for the attribute names defined by the user, which may contain topological changes. The topological changes include the number of edges and end points that are needed to resolve the geometry. The input file used in this thesis is shown in List 4-2.

```

H:\ tm_assy.prt
tm_assy.exp
hlod 1 zcrv
hlid 1 zcrv
per1 8 zcper
per2 8 zcper

```

List 4-2 Input file for geometric processor

The first line tells the geometric processor which CAD model to open. The second line indicated the name of the expression file to use to update the model. The rest of the lines define the families to check, mesh size to assign to the curves, and the name of the families to assign to the curves. The commands used to open the part and update it with the specified expression file that is specified in the input file is shown in Code Frag 4-1.

```

// open part
int load_status=0;
load_status =
prt_misc.open_part
(part_file);

if (load_status == -1)
    return -1;
// import expressions
int exp_def=0;
UF_CALL(UF_MODL_import_exp
(exp_file,exp_def));

// Update the model
UF_CALL(UF_MODL_update
( ));

```

Code Frag 4-1 Code used to open the CAD model, read in the expression file and update the model

When the CAD model is updated, the new edges and points are tagged. This tagging was done using Code Frag 4-2.

```

cfd.get_objects_with_attributes(prt_misc.part);

// find the faces with the specified attributes
uf_list_p_t objs_list;
for(i=0;i<number;i++)
{
    fout << familys[i] << "\t" << i << endl;

    // Look for named sheetbodies that might cross
the periodics
    prt_misc.get_named_sheetbodies(familys[i]);

    // Check every object in the part for a specific
attribute. In our case ICCFD_FAMILY
    mdl_misc.get_named_faces
    (icem.attr_list,UF_ATTR_any,"ICCFD_FAMILY",familys[i]);

    // tag each edge
    mdl_misc.prep_objects(mdl_misc.obj_list,familys
[i],fam_size[i],obj_families[i]);
}

// create points on the edges
cfd.assign_pnts();

```

Code Frag 4-2 Find the specified family edges and add points and tag them

When the model has been processed, it was saved into the current working directory with prefix “mod_” added to the part name. Some images of ICEM input files are shown in Figure 4-15. They indicate the input files generated with and without the geometry processor. As seen in the before and after images below, curves and points were added on the outer diameter to the injection holes, mixing chamber, and plenum

Figure 4-15 With and without running the geometry processor

4.3 Grid Generation

To automate the grid generation process a Tcl script was created. This script was broken up into modules for reusability. This allows for new modules to be added by using the line “source ...” as shown in Code Frag 4-3

```

# This script will access the ug model and
# create a grid with prisms
set tetfile ug_import.tin
set meshfile tetra_mesh.tin
set denfile densvol.tin
set ug_part mod_tm_assy.prt
source "H:\mdl_master\icem_scripts\getUgModel.tcl"
source "H:\mdl_master\icem_scripts\createGrid.tcl"
if {[file exists ./meshfile]} {
    source "H:\mdl_master\icem_scripts\createGrid.tcl"
}
if {[file exists ./meshfile]} {
    source "H:\mdl_master\icem_scripts\gridSmooth.tcl"
}
exit

```

Code Frag 4-3 Script created to automate the grid generation process

The **set** command is used to set variables used in all of the modules. After the variables were set, then the UG model was imported using the getUGModel.tcl module.

Then the script calls the createGrid.tcl, which creates the grid input file called ug_import.tin. Next, a file containing the density volumes was merged with the input file and saved. These steps are shown in Code Frag 4-4

```

.
.
.
set tetfile ug_import.tin
set densfile density.tin
set ug_part mod_tm_assy.prt
ic_trans_ug_tetin $ug_part ./
$tetfile ug_default {} {}
ic_load_tetin ./ $tetfile 0 1
ic_load_tetin ./ $densfile 0 1
ic_save_tetin ./ $tetfile 0
.
.
.

```

Code Frag 4-4 Code to import geometry from Unigraphics and add density volumes

After the tetin file was created, the grid was created and smoothed using the following command:

```
ic_run_tetra ./tetfile ./tetra_mesh.uns run_cutter 1 run_smoother 0  
smoother_n_iter 0 smoother_aspect 0.4 n_processors 2 log tetra_cmd0.log
```

This command reads in the ug_import.tin file, creates a mesh called tetra_mesh.uns, turns the smoother operations off, and specifies the uses of two processors. The tetra_mesh.uns file was then smoothed using the following command

```
ic_smooth_elements upto 0.3 metric Quality iterations 10  
prism_warp_weight 0.5 no_collapse 1 smooth TETRA_4 smooth TRI_3
```

This selects any tri_3 of tetra_4 elements under a quality of .3 to undergo up to 10 smoothing iterations until the element quality exceeds .3.

When the model is smoothed it will be saved for visual inspection by using the following command

```
ic_save_unstruct ./tetra_mesh.uns 1 {} {} {} {} 0
```

Visual inspections were needed to ensure that the geometry was imported correctly and the resulting grid was valid. After this the valid grids were export to a PATRAN neutral file format for analysis.

4.4 CFD Analysis

The analysis phase consists of specifying the boundary conditions, preprocessing the grid generated in ICEM, analyzing the model, and postprocessing the solution to visualize the results.

4.4.1 Preprocessing

For the cases in this study, preprocessing is needed to read in the grid from ICEM, which converts it to a model file that can be used by the solver. The model file consists of the grid, with the boundary condition properly assigned to the necessary nodes. After the

model file is created, it is broken up for parallel processing. At this point the model is ready to have boundary conditions specified.

4.4.2 Boundary Conditions

For the two stream mixing problem, the mass flows of the two streams were specified. The hot stream flow rate was set to 0.033 lbm/s at 500 °F. The cold stream flow rate was set at 0.066 lbm/s and 70 °F. The desired pressure drop across the liners was set to be 3%. The table used to specify the boundary conditions (BC's) can be seen in Appendix A. To determine the flow velocity into the plenums, equation 4-2 was used:

$$V = \frac{\dot{m}}{\rho A} \quad 0-2$$

where V is the velocity, m dot is the mass flow rate, ρ is the density of the fluid, and A is the flow area. Using this equation the mass flow rate for the first and second streams was calculated. The velocities calculated for the different plenums can be seen in Figure 4-16

Figure 4-16 Flow velocities into the plenums.

With the boundary conditions specified the next step is to solve the model.

4.4.3 Analyzing the Model

To arrive at a solution, the solver iterated until the parameters converged to their final values. It was determined that it required around 2000 iterations for the solution to converge. When converged, the solution was checked to verify that the flow field had converged. Next, each case was post-processed to view the results.

4.4.4 Postprocess the Results

The postprocessing of each case was done using an Enight command file. The command file was a macro file that was generated by interactively recording the steps used to postprocess the case. The file was then edited to repeat certain commands at different locations, calculate variables, and generate images of the flow field for the desired variables. For each case, the script first opens the model and reads in the results. This was done using Code Frag 4-5.

```
data: path H:\thesis001\cfd\  
data: geometry model.db  
data: result results.db-I002601  
data: read
```

Code Frag 4-5 Enight commands to read in the model and results

When the model is read in, different screen grabs are taken to display the model and the grid. Next, a pressure map down the centerline of the model was created. This provided a visual picture of the pressure field as a sanity check. Next, the script took point measurements in the flow field to find the pressures that were used to calculate the pressure drop. When the pressure portion of the script was complete, the script generated temperature slices of the flow field at different locations to visualize the mixing of the

flow field. At each slice, a maximum and a minimum temperature were recorded. Finally, the script wrote all of the calculated scalar variables into an output file.

4.5 Design Review Tool

Upon completion of the post processing of a case, the Design Review Tool (DRT) was employed to document each solution. It provided a standard method to review each solution. The DRT is a Python script, used to generate Hyper Text Markup Language (HTML) pages. The HTML pages document each case.

The DRT script was composed using different classes to provide reusability. To set up the DRT, certain variables were modified to indicate the location of the study directory and the prefix used for each CFD case. The lines in the DRT that were updated for this project are shown in Code Frag 4-6.

```
in_path = 'H:\\prelim_cfd_study\\www\\'  
out_path = 'H:\\prelim_cfd_study\\www\\design_review\\'  
out_index = 'index.html'  
dir_prefix = 'thesis0'
```

Code Frag 4-6 Input needed to define the location of the files for the DRT

The *in_path* specifies where the solutions were located, *out_path* indicates the directories where the solutions are copied to, *out_index* is the name used for the index page, and *dir_prefix* indicates the prefix of the solution directories. With these variables updated to reflect the location of the files, the Design Review is ready to be run.

To allow for code reusability, each section used on the case page was created using separate modules known as classes. The model class used to create the model section is shown in Code Frag 4-7. It takes the file *model.jpg* created in the Enight script, along with the expression file used to define the UG model and displays them in a

web page. If the user wanted to display more information or change the formatting, all that is required is to modify the model class and rerun the DRT. The other modules were set up in a similar manner.

```
class model_section:
    def __init__(self,folder,exp_file,img_dir):
        self.indir = ''
        self.outdir = ''
        self.folder = folder
        self.params = exp_file
        self.image = self.outdir+'\\'+img+dir+'\\model.jpg'
        self.table_list = []
        self.table_list1 = []
        self.doc = HTMLgen.SimpleDocument(title="Model")
        self.param_table = HTMLgen.Table(border=1, width =
            80,cell_align="left", heading=['Model '+self.folder[-
            3:]+'
            Info'],heading_align='center',heading_valign='middle'
            ,column1_align='center',colspan=[2,1],cell_line_break
            s=1)

    def get_params(self,path):
        p_file = open(path+'\\'+self.params)
        p_data = p_file.readlines()
        p_file.close()

        lines = string.join(p_data)
        self.param_table.body = []
        self.param_table.body.append([HTMLgen.Image ('img_dir
            +'\\iso001.jpg', width='480',height='360'),lines])

        self.doc.append(self.param_table)
        self.doc.write(self.outdir+'\\model.html')
```

Code Frag 4-7 Model class module for the DRT

The first web page created by the DRT displays a listing of all of the cases with the predicted temperature spread at the exit plane as seen in Figure 4-17.

Case #	Delta T
<u>01</u>	40.484
<u>02</u>	21.041
<u>03</u>	18.022
<u>04</u>	91.509
<u>05</u>	38.593
<u>06</u>	72.38
<u>07</u>	97.768
<u>08</u>	31.94
<u>09</u>	68.642
10	---

Figure 4-17 A screen grab of the preliminary CFD study page

From the “Two Stream Mixing Study” page, the design team can see which cases are completed with the objective value and which cases are in progress. The finished cases are selectable and their results can viewed by selecting a case number. The cases that are in progress display a “---” in the objective column and are not selectable. By selecting a case number for a completed case, the case page is displayed. This page provides a quick glance at the different sections of the case as seen in Figure 4-18

Figure 4-18 DRT: Case template

Each section can be reviewed by clicking on an image. This allows the user to review the model and solution. The model and grid section provides a review of the geometry and the grid. The model section displays a wire frame view of the model, along with the expression file used to update the CAD model. The page that is displayed when the model image is clicked is shown in Figure 4-19. The grid section shows a slice of the grid, with a table listing the families and elements that are tagged as boundary conditions. The page that displays when the grid image is clicked is shown in Figure 4-20

Figure 4-19 DRT: Model section

Figure 4-20 DRT: Grid section

The next two sections, flow conditions and pressure map, help verify that the CFD solution was run properly. The flow condition page, shown in Figure 4-21, provides

a visual check of the flow conditions. While the pressure map page, shown in Figure 4-22, provides a visual inspection of the pressure field.

Figure 4-21 DRT: Flow conditions section

Figure 4-22 DRT: Pressure map section

The final two sections, flow results and objectives, provide a visual representation of the results for each case. In the flow results section, different variables can be reviewed. For this thesis, only two were look at, however, if more images were created they could easily be added to this section by modifying the flow results module used by the DRT.

The page that is displayed when the flow results section is clicked on can be seen in Figure 4-23. From the flow results section page the different parameters are shown and can be reviewed by clicking on the image. When the temperature image is clicked on, all of the temperature images generated are displayed. A screen grab of the temperature page is shown in Figure 4-24. At the top center of each result page is an image that shows the axis for the different slices. The top center image for the temperature results can be seen in Figure 4-25.

Figure 4-23 DRT: Flow results section

Figure 4-24 DRT: Temperature results page

Figure 4-25 DRT: Temperature slice information image

The objective section, shown in Figure 4-26, provides a visual representation of the how quick the two streams are mixing and the temperature ranges of the flow as it

progresses downstream. On the left hand side of the page, Figure 4-25 is shown to provide a reference of where the temperatures were calculated. On the right hand side of the page a graph is displayed to shown the maximum and minimum temperature in the flow field. By clicking on the graph an enlarged image of the graph is displayed.

Figure 4-26 DRT: Objective section

DISCUSSION OF RESULTS

This section presents the results of the CAD model, geometry processor, and the grid generation as applied to the conceptual study of the two stream mixing problem. The application of the method resulted in a process that could be used to analytically screen many different concepts quickly and efficiently. The results of the grid automation process and the Design Review Tool (DRT) are also presented.

To better understand the work done in this thesis to automate the grid generation process one must compare it to the current process in industry. As discussed in the introduction, this process is time consuming, costly, and involves multiple people. Automation can help eliminate these problems.

5.1 Results: Automated Grid Generation of Parametric Models

Automating the CAD-to-Grid process for parametric models was aided by the creation of the Master Cross Sectional Sketches (MCSS) and component modeling techniques. When these steps were added, the air solid assembly came together quickly and allowed for quick interchangeability of components. A standard process was then captured in a script to generate the grid for each new configuration. The following sections will discuss how this method improves on the current process.

5.1.1 Results: MCSS

The MCSS was key in defining the critical assembly requirements. These requirements being the injection point of the two mixing streams, the mixing chamber height and length. Creating the MCSS required around an hour, but provided the parametric framework for the different concepts. The MCSS ensured that, each component in the assembly aligned properly. This approach added an extra step to the standard practices used today, but reduced the time required to assemble each component to around 15 seconds and ensured they were interchangeable.

5.1.2 Results: Component Modeling

Each concept was constructed using geometric entities referenced from the MCSS. This allowed for the concepts to have common entities and datums. In addition, tagging at the component level for the grid generation phase made it possible to interchange them in the assembly, which removed and added the necessary tags for the grid input file. When compared with the standard practice this takes upfront thought and time, but the payoff comes in the interchangeability of components in this and other potential assembly models.

5.1.3 Results: Assembly Model

Using the MCSS as a framework, the assembly model was able to swap in and out the different front ends and liners. The assembly model used to generate all of the different cases is shown in Figure 5-1. Note the components shown in Figure 5-1 are front-end concept #1 with triangular holes in the liners that are located off of the periodic

surfaces. Selected cases were run, which show the flexibility of the model, with different front-end concepts and hole patterns arrangements as shown in Figure 5-2 and Figure 5-3.

Figure 5-1 Base assembly with front-end concept #1 and liner hole concept #3

Figure 5-2 Case 002 air solid assembly with front-end concept #2 and liner hole concept #1

Figure 5-3 Case 006 air solid assembly with front-end concept #3 and liner hole concept #2

Contrast this with today's standard process of placing the different components in an assembly with the proper mating conditions. Today's approach often requires creative solutions to remove any slivers created from misalignment of the different components in the assembly. These solutions are generally done quickly to fix the model with little thought of reusability. Therefore, when changes are made down to the assembly, one or more features would likely fail.

Each concept similar to the ones above was processed using the geometry processor. This ensured that specified families in the input file were tagged and saved in the proper directory. All of the nine processed models resulted in valid input files for ICEM without any human intervention. Had this been done using standard practices, it would have required four to eight hours to modify and tag the nine models for ICEM vs the 2 minutes per model required for the geometric processor.

5.1.4 Grid Generation

Using the scripting capabilities of ICEMCFD the grid generation step was automated. The script opened up the CAD model and extracted the tagged geometry. Each input file was then loaded into ICEMCFD. The input file that was automatically generated and imported into ICEM for case004 is shown in Figure 5-4.

Figure 5-4 ICEM screen capture of the input file that was imported from Unigraphics

From the input file a grid was then created and smoothed using the script. A slice through the center of the grid that was generated automatically is shown in Figure 5-5.

Figure 5-5 The finished grid that was generated for case004

To import the geometry, generate the grid, smooth it, and save it for inspection took around fifteen minutes. Standard practices would have required at least five to ten more minutes along with a human in the loop. Past experience has shown that it is necessary to check each grid to verify that the grid is error free of non-manifold elements, uncovered faces, low quality elements, etc. However, for the simplified geometry looked at in this thesis, all of the nine grids generated were error free.

To develop the tools used in this thesis took around 400-500 hours. However, once they are developed, they can be applied to other studies with little additional development time. In Table 5-1 is shown a comparison between the current process and the one used in this thesis. As shown in Table 5-1 the new approach takes a bit more time in the modeling set up for the first model than the current approach. However, it took ~ 40 % of the current approach to generate nine models and review them.

Table 5-1 Current approach vs New approach

Current Practice			New Approach	
Steps	First Model	Next 8 Models	Single Model	Next 8 Models
Modelling	8-12 hrs	32-36 hrs	12-16 hrs	4-6 hrs
Grid Generation	8-12 hrs	32-36 hrs	8-12 hrs	3 hrs
Solution time	16 hrs	36* hrs	16 hrs	12** hrs
Report Results	4-8 hrs	32-56 hrs	4-8 hrs	3 hrs
Totals	36-48 hrs	132-164 hrs	40-52 hrs	22-24 hrs
Totals for nine solutions		168-212 hrs		62-76 hrs
* Assumes that three solutions were run at a time due to the added time to create a model				
** All eight solutions were run together due to the decreased time to generate a model				

5.2 Results: Design Review Tool

The intent of the DRT was to minimize the time it takes to document each solution. Currently, the standard practice in industry is to post process the CFD solution and generate images for a set of parameters. The analyst then proceeds to make a presentation. The presentation consists of a discussion of the model, a review of the boundary conditions, and pressure field, and then a review of the results such as temperature, velocities, etc. If performed manually, the time it takes to document each case can take hours. When iterations of the model are performed, the analyst commonly takes the previous presentation as a template and updates it. This process of updating the presentation results in delaying the review of the results until a formalized presentation can be made.

The DRT helped cut this time down considerably. To determine the effectiveness of the DRT the following questions are discussed. First, does the DRT present the results effectively? Second, are the results presented in a timely manner? Third, are results of each case setup to be archived for later use?

5.2.1 Does the DRT present the results effectively?

The DRT provides a standard format for each case which was set up in a way that demonstrates the elements that are reviewed in industry today. This format consists of standard layout for each case. This layout was shown previously in Figure 4-18 through Figure 4-26. By using a standard format each case can be easily compared to the other cases in the study. This format can be changed as needed by the design team and quickly updated by modifying the DRT script and reprocessing the data.

5.2.2 Are the results presented in a timely manner?

When each case is completed, the DRT is executed. It takes around 2.5 seconds to create all of the web pages, organize the images, and create the graphs necessary to document each case. For cases with more results and images the time required to document the case would take longer. When the DRT is finished running, each completed solution was ready for review by the design team. The standard practice used in industry today ranges from one hour to a day to document each solution. The time savings of the DRT provided the engineer and design team more time to review the results and less time generating and formatting the results for review.

5.2.3 Are results of each case setup to be archived for later use?

The DRT uses a common directory structure. This directory structure places the results of the different cases in a common directory. The base directory has a folder for each case. Each case folder has their results placed in this folder as well as an images folder, that contains all of the images. This enables the DRT to create web pages using a relative directory structure for each case. The relative directory structure allowed the base

directory to be archived and retrieved in a different directory while maintaining the functionality of the index page.

To test out the archiving and retrieving capabilities of the tool the base directory was compressed and moved to hypothetical archival location. The study directory was then retrieved and moved to a different location and uncompressed. When the index page was opened, the index page worked as designed.

CONCLUSIONS

The objective of this thesis was to show that: 1) Valid CFD grids be made in an automated way from parametric models that undergo predictable topology changes? 2) Valid CFD grids be made in an automated way when one concept is replaced with a different concept in an assembly, 3) CFD data can be evaluated more efficiently for design reviews, and 4) CFD data can be effectively documented for later use?

To achieve the first two objectives a method for going from the CAD model to a valid grid was developed which used a geometric processor to handle the predetermined topology changes. This method showed that:

- using Master Cross Sectional Sketches (MCSS) provides a common framework for the components and assembly,
- components can be tagged for analysis at the component level and used at the assembly level,
- a geometry processor can successfully update the necessary tags in a parametric models with predetermined topology changes,
- valid grids can be generated for CFD in an automated way, and
- more concepts can be evaluate in a timely manner

To demonstrate the last two objectives a DRT was created. This tool proved that:

- by using a standard file structure, and file naming convention each case can be formatted for review in under 10 seconds.
- a modular structure allows sections to be modified, added, or removed in order to meet the needs of the design team, and
- the results can be archived for later use when the study is completed.

The method and tools presented in this thesis were formally verified using three different front-end and liner concepts for a two stream mixing problem. A total of nine different grids were generated to evaluate the temperature range of the two streams at different locations in the mixing chamber. The results of the method and tools were documented in the previous chapters.

It was found that by using the method outlined in this thesis that parametric models with predetermined topology changes as well as component replacement could be used to create valid grid in under an hour, where it currently takes anywhere from two hours to a week in industry. The general approach presented here will allow the aerospace, automotive, and other industries to evaluate more concepts before one concept is selected for the detailed design phase. It will also help them look at different features in the design space once the detailed design phase has begun.

The DRT was shown to be useful to present the data of nine cases for review in less than a minute and in a consistent format. It enabled the CFD analyst and design team to focus more time on the results and less time on the preparation. In addition, the data can be compared in a more consistent manner by using a standard format to review the results. This work also provides a starting point for future work in coupling the CAD tool more closely with CAE analysis tools.

6.1 Future Work

There are multiple activities that would take the method outlined for this thesis to the next level. One activity would be to integrate the geometry processor, which handled predetermined topology changes (i.e. Holes on the periodics, component interchanging, new edges created on a surface) with the discrete feature work done by King (2004). Another activity would be to validate this process using more complex geometry and cases from industry.

As for the DRT more work is needed to:

1. extend the capabilities of the tool for comparing the different concepts (i.e. allowing the user to see a side by side comparison of two or more case of their choice),
2. make it more user friendly by adding a graphical user interface, and
3. make it more robust to adapt to different conceptual studies and analyses.

By focusing on these areas of the DRT it will become a vital tool in the design process and help in the review and documentation of each case run in a conceptual study.

REFERENCES:

- Alonso, Juan J., Martins, Joaquim R. R. A., Reuther, James J., Haimes R., Crawford, Curran A. (2003) "High Fidelity Aero-Structural Design Using a Parametric CAD-Based Model" *AIAA* 2003-4329.
- Anderl, R. and Mendgen, R. (1995) "Parametric Design and its Impact on Solid Modeling Applications" *Solid Modeling*, Salt Lake City, Utah, pp 1-12
- Ault, Holly K. (1999) "Using Geometric Constraints to Capture Design Intent" *Journal for Geometry and Graphics*, vol 3, No. 1, pp 39-45.
- Crocker, D. Scott, Nickolaus, Dan, Smith, Clifford E., (1998) "CFD Modeling of a Gas Turbine Combustor from Compressor Exit to Turbine Inlet" *International Gas Turbine & Aeroengine Congress & Exhibition*, Stockholm, Sweden, June 2-5
- Cervenka, Michael, (2000) "The Rolls-Royce Trent Engine", <http://www.msm.cam.ac.uk/phase-trans/mphil/Trent1/sld014.htm>
- Dawes, W.N., Dhanasekaran, P.C., Demargne, A.A.J., Kellar, W.P., and Savill, A.M., (2001), "Reducing Bottlenecks in the CAD-to-Mesh-to-Solution Cycle Time to Allow CFD to Participate in Design," *Journal of Turbomachinery*, Vol. 123, pp. 552-557.
- Haimes, R., and Crawford, C., (2003), "Unified Geometry Access for Analysis and Design" *Proceedings, 12th International Meshing Roundtable*, Santa Fe, New Mexico, Sept 14-17, 2003, pp. 21-31.
- Hardee, E., Chang, Kuang-Hua, Tu, Jian, Choi, Kyung K., Grindeanu, I., (1999) "A CAD-Based design parameterization for shape optimization of elastic solids" *Advances in Engineering Software*, vol 30, pp 185-199.
- Hoffmann, C.M., Kim, K.J., (2001), Towards valid parametric CAD models, *CAD Computer Aided Design*, Vol 33, Issue 1, pp 81-90.
- Hogge, D., (2002), *Integrating Commercial CAx Software to Perform Multidisciplinary Design Optimization*, M.S. Thesis, Brigham Young University.
- King, M., (2004), *A CAD-centric approach to CFD analysis with discrete features*, M.S. Thesis, Brigham Young University.

- Lawson, John, Erjavec, John, (2001) *Modern Statistics for Engineering and Quality Improvement*, Wadsworth Group, Duxbury.
- Lee, K., (1999), *Principles of CAD/CAM/CAE Systems*, Addison Wesley, Reading, MA.
- Malecki, Robert E., Rhie, Chae M., McKinney, Randal G., Ouyang, Hong, Syed, Saadat A. Colket, Meredith B., Madabhushi, Ravi K. (2001) "Application of an Advanced CFD-Based Analysis System to the PW6000 Combustor to Optimize Exit Temperature Distribution – Part 1: Description and Validation of the Analysis Tool" *Proceedings of ASME: ASME TURBOEXPO 2001*, New Orleans, Louisiana, June 4-7, 2001-GT-0062.
- Newman III, J. C., Taylor III, A. C., Barnwell, R. C., Newman, P. A., Hou, G., (1999) "Overview of Sensitivity Analysis and Shape Optimization for Complex Aerodynamic Configurations", *Journal of Aircraft*, vol 36., No 1. January-February
- Rohm III, T., (2001) Graphical Creation of CAD Parametric Application Programs, M.S. Thesis, Brigham Young University.
- Rohm III, T., Jones, C.,, Tucher, S., Jensen, C., (2000), "Parametric Engineering Design Tools and Applications," ASME Design Automation Conference, Baltimore, MD, September, 10-13, 2000, pp.657-664.
- Samareh, J. A., (1999) "Status and Future of Geometry Modeling and Grid Generation for Design and Optimization," *Journal of Aircraft*, vol. 36, pp. 97-104.
- Smith, R.J., Leshchziner, M.A., (1996), *Automatic grid generation for complex geometries*, Aeronautical Journal, Vol. 100, Issue 991, pp. 7-14.
- Srinivasan, H., Jensen, C. G., Kopper, F. C., Staubach, J. B., and Pack, D. R., "Assembly Parametrics for Multidisciplinary Design Optimization" 8th *International Society for Productivity Enhancement Conference on Concurrent Engineering*, Anaheim, CA, July 29 – August 1, 2001, pp. 158-165
- Subrahmanyam, S., DeVries W., and Pratt, M., (1995), "Feature Attributes and their Role in Product Modeling," *Proceedings of the Symposium on Solid Modeling and Applications*, Salt Lake City, UT, May 17-19, 1995 pp. 115-124.
- Tangirala, Venkat E., Tolpadi, Anil K., Danis, Al M., Mongia, Hukam, (2000) "Parametric Modeling Approach to Gas Turbine Combustor Design" *Proceedings of ASME: ASME TURBOEXPO 2000*, May 8-11 Munich Germany, 2000-GT-129

Ulrich, Karl., Epinge, Steven., (2000) *Product Design and Development*, McGraw-Hill.

APPENDIX

Appendix A: Setup information and flow calculations

	Inlet	OD	ID		Total
Flow Split	33.3%	33.3%	33.3%		100.0%
	0.033	0.033	0.033		0.1 lbm/s
Pres Amb	14.700				
Pres Drop	0.441				
Mdot	0.1 lbm/s				
Ps1	15.141 psia				2180.304 psfa
Ps2	15.141 psia				2180.304 psfa
T1	500 F				961 R
T2	70 F				531 R
rho1	2.46102E-05 lbm/in ³				0.001321765 slugs/ft ³
rho2	4.45394E-05 lbm/in ³				0.00239212 slugs/ft ³

Boundary Conditions						
BC Name	BC Tag	BC Type	Area	P	Rho	Vel (ft/s)
FE Inlet	1	inlet-speed	10.282	15.1410	2.46E-05	10.9780
Exit	2	exit	13.7090	14.7000	3.79E-05	16.0404
Periodic	11	periodic-a				
Periodic	12	periodic-b				
ID Inlet	20	inlet-speed	5.0840	15.1410	4.45E-05	12.2673
OD Inlet	21	inlet-speed	8.6250	15.1410	4.45E-05	7.2309
Wall	600	moving-wall				
Wall	620	moving-wall				
Wall	630	moving-wall				
Wall	640	moving-wall				

ACd = measured equivalent area of the burner liner (in²)

g = unit conversion factor, 32.2 lbm*ft/lbf*s²

Delta P = measured pressure drop across the combustor liner (psid)

T = Combustor inlet temperature, deg F

P = Static pressure at the combustor liner inlet, psia

mdot = $\sqrt{2 \cdot 32.2 / 144} \cdot ACd \cdot \sqrt{\rho \cdot \Delta P}$ (lbm/s)

$\rho = 2.698825 \cdot P / (T + 459.67)$ (lbm/ft³)

Flow Area Calculations				
	OD	ID	FE	Units
ACd	0.271	0.271	0.364	(in ²)
delP	0.441	0.441	0.441	psi
Rho	0.077	0.077	0.043	lbm/ft ³
Mdot	0.033	0.033	0.033	lbm/s

Initial Guesses For Discharge Coefficient			
	Circle	Ellipse	Triangle
Cd	0.7	0.65	0.6
Geom Area	0.387	0.416	0.451

	4 Slots	Cylinder	4 Cylinders
Cd	0.75	0.8	0.95
Geom Area	0.485	0.455	0.383

Concept 1 ACD calculations			
Target ACd	FE	Geom Area	Cd
0.33	0.307	0.357	0.860
0.364	0.339	0.394	0.861
0.4	0.370	0.432	0.856
			0.86
Target ACd	Liner	Geom Area	Cd
0.24	0.246	0.324	0.757
0.271	0.273	0.382	0.714
0.3	0.294	0.423	0.694

Concept 2 ACD calculations			
Target ACd	FE	Geom Area	Cd
0.33	0.354	0.384	0.922
0.364	0.354	0.384	0.922
0.4	0.354	0.384	0.922
Target ACd	Liner	Geom Area	Cd
0.24	0.279	0.369	0.755
0.271	0.304	0.417	0.728
0.3	0.324	0.462	0.702

Concept 3 ACD calculations			
Target ACd	FE	Geom Area	Cd
0.33	0.239	0.330	0.724
0.364	0.264	0.364	0.726
0.4	0.291	0.400	0.729
Target ACd	Liner	Geom Area	Cd
0.24	0.267	0.345	0.772
0.271	0.293	0.397	0.738
0.3	0.322	0.445	0.722

Appendix B: DRT Scripts

```
# Compare CFD cases and create web pages
import sys
import os
import string
import shutil
import time
from pylab import *
import Image
sys.path.append('D:\\python_extras\\HTMLgen')
import HTMLgen, barchart
sys.path.append('H:\\prelim_cfd_study\\mdl_master\\py_scripts')
from allstar_classes import postflowfile
from design_review_classes import *

in_path = 'H:\\prelim_cfd_study\\www\\'
out_path = 'H:\\prelim_cfd_study\\www\\design_review\\'
out_index = 'index.html'
param_file = 'tm_assy.exp'
img_dir = 'images\\'
cfd_dir = 'cfd'
dir_prefix = 'thesis0'
sect_names = ['Model', 'Grid', 'Flow Conditions', 'Pressure Map', 'Flow Results', 'Objectives Graphs']
sect_imgs = ['iso001.jpg', 'image003.jpg', 'ACd_plot.png', 'image004.jpg', 'temp000.jpg', 'T_plot.png']

# Set the timer to zero
exec_time = time.time()
start_time = exec_time

# Get each directory in the specified path
dirs = []
dir_loc = []
files = os.listdir(in_path)

# Remove items that are not directories
files.sort()
for x in range(0, len(files)):
    # Check to see if directory already exist in the out path directory
    if os.path.isdir(out_path+files[x])==1:
        junk = 1
    elif files[x][:len(dir_prefix)]==dir_prefix:
        # Copy the files from the in_path to the out_path directory
        print "Copying ", out_path+files[x]
        os.mkdir(out_path+files[x])
        if os.path.isdir(out_path+files[x]+'\\'+img_dir)==1:
            shutil.copytree(in_path+files[x]+'\\'+img_dir, out_path+files[x]+'\\'+img_dir)
        print "-----"+in_path+files[x]+'\\'+param_file
        if os.path.isfile(in_path+files[x]+'\\'+param_file)==1:
            shutil.copy(in_path+files[x]+'\\'+param_file, out_path+files[x]+'\\'+param_file)

    if os.path.isdir(out_path+files[x])!=True or files[x][:len(dir_prefix)]!=dir_prefix:
        #print 'Removing '+files[x]+' from the list'
        dir_loc.append(files[x])

for x in dir_loc:
```

```

files.remove(x)

for x in range(0,len(files)):
    exec_time = time.time()
    if os.path.isdir(out_path+files[x]):

        # Gather the data from the allstar header file
        ci = case_info(in_path+files[x]+'\\'+cfd_dir+'\\')
        ci.get_info()

        # Create indexes of the results
        if os.path.isdir(out_path+files[x]+'\\'+img_dir)==1:

            case_doc = HTMLgen.SimpleDocument(title=files[x])
            case_table1 = HTMLgen.Table(border=0, width=600, cell_align="center",
            tabletitle='Configuration '+files[x][-
            3:],heading_align='center',heading_valign='middle',column1_align='center')#,colspan=[2,1])
            case_table = HTMLgen.Table(border=0, cell_align="center", heading=['Configuration '+files[x][-
            3:]+' Case Info'])
            case_table.body.append(['Iterations: '+str(ci.iterations)])
            case_table.body.append(['Elements: '+str(ci.elements)])
            case_table.body.append([HTMLgen.Href(param_file,"Parameters")])

            #####
            # Make the pages for the links for the case info page
            #####
            ms = model_section(files[x],param_file,img_dir,sect_names[0])
            ms.outdir = out_path+files[x]
            ms.get_params(in_path+files[x]+'\\')
            gd = grid_section(files[x],img_dir,sect_names[1])
            gd.outdir = out_path+files[x]
            gd.get_params(in_path+files[x]+'\\')
            bcs = check_bcs(files[x],img_dir,sect_names[2])
            bcs.make_page(in_path+files[x]+'\\'+cfd_dir+'\\',out_path+files[x]+'\\')
            pres_res = pres_results(files[x],img_dir,sect_names[3])
            pres_res.make_page(out_path)
            flow_res = flow_results(files[x]+'\\',out_path+files[x]+'\\',img_dir,sect_names[4])
            flow_res.make_page(out_path)
            obj = objective(files[x],img_dir,sect_names[5])
            obj.make_page(out_path+files[x]+'\\',out_path+files[x]+'\\')

            table1_list = []

            len_sect = len(sect_names)
            for xx in range(0,len_sect,2):
                if xx < len_sect-1:

                    table1_list.append([HTMLgen.Href(sect_names[xx]+'\\.html',HTMLgen.Image(img_dir+'/' +sect_imgs[x
                    x],width='300',height='225')),HTMLgen.Href(sect_names[xx+1]+'\\.html',HTMLgen.Image(img_dir+'/'
                    +sect_imgs[xx+1],width='300',height='225'))])
                    table1_list.append([sect_names[xx],sect_names[xx+1]])
                else:

                    table1_list.append([HTMLgen.Href(sect_names[xx]+'\\.html',HTMLgen.Image(img_dir+'/' +sect_imgs[x
                    x],width='300',height='225'))])
                    table1_list.append([sect_names[xx]])

```

```

case_table1.body = table1_list
case_doc.append(case_table1)

filename = out_path+files[x]+'\\index.html'
case_doc.write(filename)

dirs.append([HTMLgen.Href(files[x]+'/'+'index.html',files[x][len(dir_prefix):]),obj.Tdelta[-1]])
else:
    dirs.append([files[x][len(dir_prefix):],'---'])
exec_time = time.time() - exec_time;
print "Time: ",exec_time

doc = HTMLgen.SimpleDocument(title="CFD Study")

head_list = ["Case #","Delta T"]
table = HTMLgen.Table( tabletitle="Two Stream Mixing Study", border=1, width=500,
cell_align="center",
heading=head_list,heading_align='center',heading_valign='middle',column1_align='center')

doc.append(table)

table_list = []
for x in dirs:
    table_list.append(x)

table.body = table_list
os.chdir(out_path)
doc.write(out_index)
print "Total time: ",time.time()-start_time

```

Modules used by the DRT

Compare CFD cases and create web pages

```
import sys
import os
import string
import shutil
import time
from pylab import *
import Image
sys.path.append('D:\\python_extras\\HTMLgen')
import HTMLgen,barchart
sys.path.append('H:\\prelim_cfd_study\\mdl_master\\py_scripts')
from allstar_classes import postflowfile

class case_info:
    def __init__(self,path):
        self.dir = path
        self.header = path+"\\allstar_header.db"
        self.iterations = 0
        self.elements = 0

    def get_info(self):
        # Find the most recent allstar header file

        if os.path.isdir(self.dir):
            contents = os.listdir(self.dir)
            #print self.dir
            header_files = []
            for file in contents:
                if string.find(file,'allstar_header.db-')!=-1:
                    header_files.append(int(file[-6:]))

            if len(header_files) > 0:
                self.iterations = max(header_files)
                self.header = self.dir+"\\allstar_header.db-I%06d"%(self.iterations)

            in_file = open(self.header)
            data = in_file.readlines()
            in_file.close()

            for x in data:
                if self.elements == 0 and string.find(x,'number_of_elements_h')!=-1:
                    data = string.split(x,'=')
                    self.elements = data[1][-2]
                    #print "Elements = ",self.elements
                    #print "Iterations = ",self.iterations

class model_section:
    def __init__(self,folder,exp_file,img_dir,file_name):
        self.indir = "
        self.outdir = "
        self.folder = folder
        self.params = exp_file
        self.img_dir = img_dir
        self.image = self.outdir+"\\'+img_dir+'model.jpg'
```

```

self.table_list = []
self.table_list1 = []
self.doc = HTMLgen.SimpleDocument(title="Model")
self.param_table = HTMLgen.Table(border=1, width = 80,cell_align="left", heading=['Model
'+self.folder[-3:]+'
Info'],heading_align='center',heading_valign='middle',column1_align='center',colspan=[2,1],cell_line_
breaks=1)
self.file_name = file_name

def get_params(self,path):
    #print path
    #print self.params
    p_file = open(path+'\\'+self.params)
    p_data = p_file.readlines()
    p_file.close()

    lines = string.join(p_data)
    self.param_table.body = []

self.param_table.body.append([HTMLgen.Image(self.img_dir+'iso001.jpg',width='480',height='360'),li
nes])

self.doc.append(self.param_table)
self.doc.write(self.outdir+'\\'+self.file_name+'.html')

class grid_section:
def __init__(self,folder,img_dir,file_name):
    self.folder = folder
    self.img_dir = img_dir
    self.images = [self.img_dir+'image003.jpg']
    self.doc = HTMLgen.SimpleDocument(title="Grid")
    self.table = HTMLgen.Table(border=1,cell_align="left", heading=['Grid '+self.folder[-3:]+'
Info'],heading_align='center',heading_valign='middle',column1_align='center',colspan=[2,1],cell_line_
breaks=1)
    self.table.body = []
    self.bc_table = HTMLgen.Table(border=0,cell_align="right", heading=[HTMLgen.Heading(3,
'Boundary Conditions',
align='right')],heading_align='center',heading_valign='middle',column1_align='right',colspan=[2,1])
    self.bc_table.body = []
    self.file = '\\cfd\\allstar_flags.db'
    self.outdir = "
    self.bcs = []
    self.vols = []
    self.file_name = file_name

def get_params(self,path):
    #print path
    #print self.file
    p_file = open(path+self.file)
    p_data = p_file.readlines()
    p_file.close()

    for line in p_data:
        if string.find(line,'boundary condition') !=-1:
            bc_data = string.split(line)
            self.bcs.append(['%4d'%(int(bc_data[0])), '%d'%(int(bc_data[-2]))])

```

```

        if string.find(line,'volume') !=-1:
            bc_data = string.split(line)
            self.vols.append(['%4d'%(int(bc_data[0])), '%d'%(int(bc_data[-2]))])
        self.bcs.reverse()
        self.bcs.insert(0,[ HTMLgen.Heading(4, 'Flag' , align='right'),HTMLgen.Heading(4, 'Elements' ,
align='right')])
        #print self.bcs
        self.bc_table.body = self.bcs
        self.bc_table.body.append([HTMLgen.Heading(4, 'Volume' , align='right'),HTMLgen.Heading(4,
'Elements' , align='right')])
        for vol in self.vols:
            self.bc_table.body.append(vol)

        self.table.body.append([HTMLgen.Image(self.images[0],width='480',height='360'),self.bc_table])
        #self.table.body.append([HTMLgen.Image(self.images[1],width='480',height='360'),'\n'])
        #self.doc.append()
        self.doc.append(self.table)
        self.doc.write(self.outdir+'\\'+self.file_name+'.html')

class check_bcs:
    def __init__(self,folder,img_dir,file_name):
        self.folder = folder
        self.img_dir = img_dir
        self.acd_act = []
        self.acd_des = [.36,.,27,.,27]
        self.names = ['FE','OD','ID']
        self.pdown = 14.7
        self.p_act = []
        self.p_des = [3.0,3.0,3.0]
        self.mdot_act = []
        self.mdot_des = [.033,.,033,.,033,.,1]
        self.out_path = "
        self.images = [img_dir+'ACd_plot.png',img_dir+'pdrop_plot.png',img_dir+'mdot_plot.png']
        self.image_names = ['ACd','Pressure Drop','Mass Flow Rate']
        self.doc = HTMLgen.SimpleDocument(title="BCS.html")
        self.table = HTMLgen.Table(border=0,cell_align="center", heading=['Flow Conditions for
'+self.folder[-
3:]],heading_align='center',heading_valign='middle',column1_align='center',colspan=[2,1],cell_line_br
eaks=1)
        self.table.body = []
        self.store_results = [[],[],[]]
        self.file_name = file_name

    def get_data(self,in_path,out_path):
        self.out_path = out_path
        in_file = open(self.out_path+'pressure_points.txt')
        in_data = in_file.readlines()
        in_file.close()

        pff = postflowfile()
        pff.read(in_path+'PROBE\\post_flow.data')

        # get the mass from the different boundary conditions
        fe_mass = pff.get_bc_mass('1')
        od_mass = pff.get_bc_mass('32')
        id_mass = pff.get_bc_mass('31')
        exit = pff.get_bc_mass('2')

```



```

self.mdot_act.append(fe_mass)
self.mdot_act.append(od_mass)
self.mdot_act.append(id_mass)
self.mdot_act.append(fe_mass+od_mass+id_mass)

# Calculate the pressure drops
in_data = in_data[2:]
cham_ps = 0.0
for x in range(0,4):
    cham_ps = cham_ps + float(string.split(in_data[x])[0])/144.0
cham_ps = cham_ps/(4.0)
#print cham_ps

od_pt = 0.0
for x in range(4,6):
    od_pt = od_pt + float(string.split(in_data[x])[1])/144.0
od_pt = od_pt/2.0

id_pt = 0.0
for x in range(6,8):
    id_pt = id_pt + float(string.split(in_data[x])[1])/144.0
id_pt = id_pt/2.0

fe_pt = 0.0
for x in range(8,10):
    fe_pt = fe_pt + float(string.split(in_data[x])[1])/144.0
fe_pt = fe_pt/2.0

self.p_act.append(100.0*(fe_pt/cham_ps-1.0))
self.p_act.append(100.0*(od_pt/cham_ps-1.0))
self.p_act.append(100.0*(id_pt/cham_ps-1.0))
#print self.p_act

#calculate the ACd for each component
fe_acd = fe_mass/(1.099*sqrt(cham_ps*(fe_pt-cham_ps)/pff.get_bc_temp(1)))
od_acd = od_mass/(1.099*sqrt(cham_ps*(od_pt-cham_ps)/pff.get_bc_temp(32)))
id_acd = id_mass/(1.099*sqrt(cham_ps*(id_pt-cham_ps)/pff.get_bc_temp(31)))

# add the acd values to the list
self.acd_act.append(fe_acd)
self.acd_act.append(od_acd)
self.acd_act.append(id_acd)

# write the values to a file
post_file = open(out_path+'bc_check.txt','w')
post_file.write("ACd Values\n-----\n")
for x in range(0,len(self.acd_act)):
    post_file.write(self.names[x]+'t'+str(self.acd_act[x])+'\n')
post_file.write("\nPressure Drop Values\n-----\n")
for x in range(0,len(self.p_act)):
    post_file.write(self.names[x]+'t'+str(self.p_act[x])+'\n')
post_file.write("\nMass Flow Values\n-----\n")
for x in range(0,len(self.names)):
    post_file.write(self.names[x]+'t'+str(self.mdot_act[x])+'\n')
post_file.close()

```

```

def acd_plot(self):
    figure(1)
    N = len(self.acd_des)
    ind = arange(N)
    width =.2
    p1 = bar(ind, self.acd_des,width,color='r')
    p2 = bar(ind+width, self.acd_act,width,color='b')
    ylabel('ACd')
    title('ACd results')
    xticks(ind+width,self.names)
    xlim(-width,len(ind))
    y_max = float('%0.1lf'%(max(self.acd_des)))*1.5
    yticks(arange(0,y_max,y_max/15.0))
    legend((p1[0],p2[0]),('Design','Actual'),shadow=True)
    savefig(self.out_path+self.img_dir+'ACd_plot')
    clf()
    #show()

def mdot_plot(self):
    figure(2)
    N = len(self.mdot_des)
    ind = arange(N)
    width =.2
    p1 = bar(ind, self.mdot_des,width,color='r')
    p2 = bar(ind+width, self.mdot_act,width,color='b')
    ylabel('Mass Flow (lbm/s)')
    title('Mass Flow Results')
    names = self.names
    names.append('EXIT')
    xticks(ind+width,self.names)
    xlim(-width,len(ind))
    y_max = float('%0.1lf'%(max(self.mdot_des)))*1.5
    yticks(arange(0,y_max,y_max/15.0))
    legend((p1[0],p2[0]),('Design','Actual'),shadow=True)
    savefig(self.out_path+self.img_dir+'mdot_plot')
    clf()
    #show()

def pdrop_plot(self):
    figure(3)
    N = len(self.p_des)
    ind = arange(N)
    width =.2
    p1 = bar(ind, self.p_des,width,color='r')
    p2 = bar(ind+width, self.p_act,width,color='b')
    ylabel('Pressure Drop(%)')
    y_max = float('%0.2lf'%(max(self.p_des)))*1.5
    ylim(0,y_max)
    title('Pressure Results')
    xticks(ind+width,self.names)
    xlim(-width,len(ind))

    yticks(arange(0,y_max,y_max/15.0))
    legend((p1[0],p2[0]),('Design','Actual'),shadow=True)
    savefig(self.out_path+self.img_dir+'pdrop_plot')
    clf()
    #show()

```

```

def make_page(self,in_path,out_path):
    self.get_data(in_path,out_path)
    self.acd_plot()
    self.mdot_plot()
    self.pdrop_plot()
    table_list = []
    img_len = len(self.images)
    for x in range(0,img_len,2):
        if x < img_len-1:
            #print self.images[x]

table_list.append([HTMLgen.Href(self.images[x],HTMLgen.Image(self.images[x],width='300',height
='225')),HTMLgen.Href(self.images[x+1],HTMLgen.Image(self.images[x+1],width='300',height='225'
))])
        table_list.append([self.image_names[x],self.image_names[x+1]])
        else:

table_list.append([HTMLgen.Href(self.images[x],HTMLgen.Image(self.images[x],width='300',height
='225'))])
        table_list.append([self.image_names[x]])
        self.table.body = table_list
        self.doc.append(self.table)
        self.doc.write(self.out_path+"\\'+self.file_name+'.html')

class objective:
def __init__ (self,folder,img_dir,file_name):
    self.folder = folder
    self.img_dir = img_dir
    self.images = [img_dir+'temp000.png',img_dir+'T_plot.png']
    #self.image_names = ['ACd','Pressure Drop','Mass Flow Rate']
    self.doc = HTMLgen.SimpleDocument(title="Objectives")
    self.table = HTMLgen.Table(border=0,cell_align="center", heading=[' Objectives: '+self.folder[-
3:]],heading_align='center',heading_valign='middle',column1_align='center',colspan=[2,1],cell_line_br
eaks=1)
    self.table.body = []
    self.outdir = "
    self.indir = "
    self.x_loc = []
    self.Tmax = []
    self.Tmin = []
    self.Tdelta = []
    self.store_results = []
    self.img_dir = img_dir
    self.file_name = file_name

def get_data(self,in_path,out_path):
    self.outdir = out_path
    self.indir = in_path

    in_file = open(in_path+self.img_dir+'constants.txt')
    in_data = in_file.readlines()
    in_file.close()

    for line in in_data:
        line = string.rstrip(line)
        if string.find(line,'Tmax') != -1:

```

```

        data = string.split(line,'=')
        self.x_loc.append(float(string.rstrip(data[0][4:])))
        self.Tmax.append(float(data[1]))
    elif string.find(line,'Tmin') != -1:
        self.Tmin.append(float(string.split(line,'=')[1]))

    for x in range(0,len(self.Tmax)):
        self.Tdelta.append(self.Tmax[x]-self.Tmin[x])

def temp_plot(self):
    figure(4)
    # T min and max plot
    subplot(211)
    plot(self.x_loc, self.Tmax,'r-o',self.x_loc, self.Tmin,'b-o')
    grid(True)
    ylabel('Temperature (R)')
    title('Two Stream Mixing Results')
    y_max = 1000.0
    y_min = 500.0
    yticks(arange(y_min,y_max,50.0))
    legend(('T max','T min'),shadow=True)

    subplot(212)
    plot(self.x_loc, self.Tdelta,'r-o')
    grid(True)
    ylabel('Delta T (R)')
    xlabel('X location')
    y_max = 400.0
    y_min = 0.0
    yticks(arange(y_min,y_max,25.0))
    #xticks(self.x_loc)
    savefig(self.outdir+self.img_dir+'T_plot')
    clf()

def make_page(self,in_path,out_path):
    self.get_data(in_path,out_path)
    self.temp_plot()
    table_list = []
    img_len = len(self.images)
    for x in range(0,img_len,2):
        if x < img_len-1:
            #print self.images[x]

    table_list.append([HTMLgen.Href(self.images[x],HTMLgen.Image(self.images[x],width='300',height
='225')),HTMLgen.Href(self.images[x+1],HTMLgen.Image(self.images[x+1],width='300',height='225'
))]
    #table_list.append([self.image_names[x],self.image_names[x+1]])
    else:

    table_list.append([HTMLgen.Href(self.images[x],HTMLgen.Image(self.images[x],width='300',height
='225'))])
    #table_list.append([self.image_names[x]])
    self.table.body = table_list
    self.doc.append(self.table)
    self.doc.write(self.outdir+'\\'+self.file_name+'.html')

```

Appendix C: DRT Screen Grabs For Concept 001

Case #	Delta T
<u>01</u>	40.484
02	21.041
03	18.022
04	91.509
05	38.593
<u>06</u>	72.38
07	97.768
08	31.94
09	68.642
10	---

Index Page

Case Review Page

Model Section

Grid Section

Flow Condition Section

Pressure Section

Flow Results Section

Temperature Results Section

Velocity Results Section

Objective Section

Temperature Plane Section

Mixing Results Section