Multiphysics Software Applications in Reverse Engineering

*W. Wang[1], K. Genc[2]
[1]University of Massachusetts Lowell, Lowell, MA, USA
[2]Simpleware, Exeter, United Kingdom

*Corresponding author: University of Massachusetts Lowell, One University Avenue, Lowell, MA 01854, USA.
wego_wang@uml.edu

Abstract: This paper will explore the applications of advanced software such as ScanIP and the +FE module (Simpleware Ltd., Exeter UK) and COMSOL multiphysics software in reinventing an existing part in the absence of original design data. It showcases the reverse engineering workflow of going from a 3D computed tomography (CT) scan of an automotive engine manifold to a COMSOL simulation ready, multi-part computational fluid dynamics (CFD) or finite element (FE) mesh of the segmented geometry. It discusses the effectiveness of applying multiphysics software in reverse engineering practice with a focus on the applicability of Simpleware and COMSOL in three-dimensional (3D) image-based modeling, structure analysis and functionality evaluation.

Keywords: Reverse Engineering, 3D Image-Based Simulation, Computational Fluid Dynamics, Finite Element

1. Introduction

During the past decade reverse engineering has become a common and acceptable practice utilized by the aftermarket suppliers to provide comparable substitutes for the OEM (Original Equipment Manufacturer) counterparts in many industries, such as the automotive industry. Recently many OEMs have also started to utilize 3D image-based reverse engineering in their development of new versions of vintage parts to shorten the turn-around time. For example, a significant amount of research time can be saved while developing a more efficient automotive clutch if reverse engineering is used to build the design data base from an existing clutch geometry.

This paper showcases the workflow of going from a 3D computed tomography (CT) scan of an automotive engine manifold to a simulation ready, multi-part computational fluid dynamics (CFD) or finite element (FE) mesh of the segmented geometry that can be directly imported into the COMSOL environment. An overview of the workflow steps are shown in Figure 1. The COMSOL multiphysics software environment is capable of facilitating all steps in the modeling and simulation processes from part defining, feature based meshing to visualization and solution analysis. However, this paper aims at the demonstration of integration of Simpleware and COMSOL in reverse engineering applications, and we will focus on the first three steps (blue), which describe the workflow of generating a simulation-ready mesh from 3D images using the ScanIP+FE softrware environment.
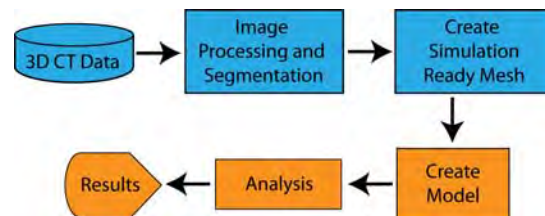

Figure1. Overview of the 3D image-based reverse engineering workflow. The input data is 3D CT scan data and is processed in ScanIP+FE (Blue) and the model creation and analysis (Orange) are processed in COMSOL.

2. Object Scanning and Image Processing

Reverse engineering a mechanical part usually starts with the reconstruction of a physical model of said part. A 3D scanner, such as a CT or microCT, or probe, is typically used for geometric data acquisition.

Data from 3D imaging modalities generally consist of a regular Cartesian grid of greyscale data representing the relative signal strength throughout the scanned volume obtained from the imaging modality [1]. In the current example, an automotive engine manifold (roughly 380 x 260 x 208mm) was scanned in a medical CT scanner with an in plane image

resolution of 0.638672mm x 0.638672mm and a slice thickness of 1mm. This stack of image data was then imported into the ScanIP environment (Figure 2).

When examining the image data both in the cross-sectional views and in the volume rendering, a certain amount of stepping and noise was observed on the manifold surface. This noise was minor and addressed by a basic Recursive Gaussian filter with Sigma filtering kernel of the size of the image resolution.

System compatibility and part performance evaluation in fit, form and function hold the key to a successful reverse engineered mechanical part. It is not unusual that apparent defects such as voids or holes will appear in a newly-reconstructed surface from the raw data. Proper follow-up editing and refinement are essential to build a high-quality part model with smooth surfaces that can be segmented for further studies such as FE/CFD analysis.
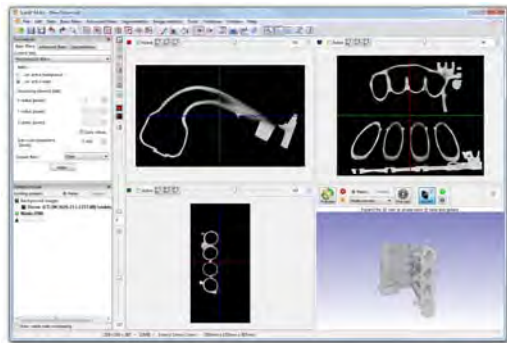

Figure 2. The ScanIP software environment showing three orthogonal views of the scanned 3D image data (grey images with black background), with a volume rendering of the entire data set (bottom right).

3. Segmentation

The most basic step of image-based simulation development and analysis that must be carried out is segmentation: that is, the identification of volumes of interest (VOI) within the image by classification of voxels into appropriate groups (eg: different parts of a scanned engine). Segmentation of 3D images is an important area of active interest and the techniques available span the gamut from manually 'painting' voxels through to use of level set and threshold methods to identify

contours as well as the use of volume rendering to aide in the visualization of the raw data (2).

Clearly, the accuracy of any resultant model will be largely dependent on the accuracy of the initial segmentation and this will be a function not just of the image resolution but a number of other factors including noise, poor contrast between parts as well as, for semi-automated and manual approaches, inter-operator differences stemming from interpretation of images.

3.1 Segmenting and Aligning the Solid Structure

In the current example, a simple threshold algorithm was used to segment the solid structure of an automotive engine manifold from the surrounding air (Figure 3). Using the tools available in ScanIP, the geometry of the manifold was then re-oriented with the image space to align the face of the manifold with the inlets to the Z-axis. This alignment was completed to aid in the selection of boundaries for the CFD mesh described below. Also, to aid with computational time, the data was resampled to 1 x 1 x 1mm.

Figure 3. Segmented geometry of the solid structure of the engine manifold from the 3D CT data.

3.2 Segmenting the Internal Fluid Structure

Using the Boolean and painting segmentation tools available in ScanIP, we were able to segment the internal regions of the manifold for CFD meshing (Figure 4).

4. Meshing the Manifold Geometries

The meshing techniques available in commercial and research codes mostly rely on using intermediate CAD (Computer Aided

Design) representations of the surface of objects of interest provided either as higher order descriptions or surface tessellations [3]. These techniques do not easily allow for more than one domain to be meshed as multiple surfaces generated are often non-conforming with gaps or overlaps at interfaces where two or more structures meet.
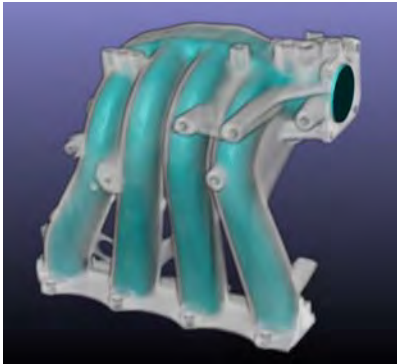

Figure 4. Segmented geometry of the internal fluid structures (blue) within the engine manifold (transparent grey).

The 'image-based approach' [1], on the contrary, is tailored to meshing from 3D imaging data, and does not decouple the surface creation from the volume meshing stage. Modeling complex topologies with possibly hundreds of disconnected domains (e.g. inclusions in a matrix), via a 'CAD approach' is virtually intractable. For the same problem, image-based meshing is, by contrast, remarkably straightforward, robust, accurate and efficient. Meshes can be generated automatically and exhibit image-based accuracy with domain boundaries of the finite element model lying exactly on the isosurfaces, taking into account partial volume effects and providing subvoxel accuracy. This meshing technique is also topology and volume preserving, avoiding loss or gain of volume and is the basis of the fully integrated +FE module in the ScanIP software environment.

Two meshing algorithms are available within the +FE module, +FE Grid and +FE Free. +FE Grid is a grid-based meshing algorithm [1]. This creates a very robust mixed hexahedral and tetrahedral mesh. +FE Free is an adaptive, fully tetrahedral meshing tool that allows more flexibility in determining mesh size and density.

Topologically complex volumes are meshed using conservative and robust algorithms whereas simpler regions can benefit from more classic tetrahedralisation methods producing a higher level of mesh density control. The result is a set of tools with a level of robustness equivalent to previous iterations of the framework that now benefit from the careful introduction of other flexible meshing techniques where possible. This flexibility allows the setting of different density zones throughout the model, effectively reducing the overall number of elements required to capture a given geometry, while allowing to increase the mesh density around areas of greater interest if necessary.

Users are also able to add and visualize element/node sets and CFD boundary conditions (BCs) such as velocity inlets and pressure outlets.

4.1 Meshing the Solid Structure

The solid structure was meshed using the +FE Free algorithm, generating a fully tetrahedral mesh of the solid volume (Figure 5).
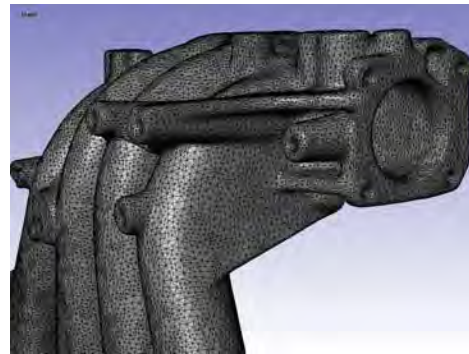

Figure 5. FE mesh of the engine manifold.

4.2 Meshing the Internal Fluid Structure

A CFD mesh of the internal fluid structure was generated using the +FE Free algorithm where the mesh quality was optimized against a skewness metric – i.e. target for all elements to be below 0.9 (Figure 6). The boundaries of the internal fluid structure were defined as velocity inlets and a pressure outlet as illustrated in Figure 6.

The COMSOL CFD module can perform multiphysics-flow simulations. Compressible as well as incompressible flows can be combined with advanced turbulence models, and forced and natural convection in analysis.
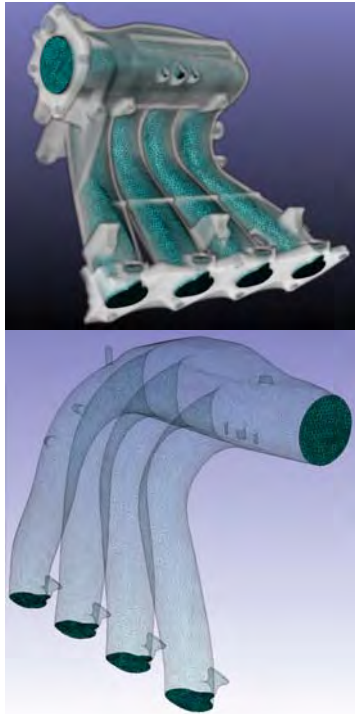

Figure 6. (Top) CFD mesh internal fluid structures (blue) within the engine manifold (transparent grey). (Bottom) Highlighted pressure outlet and multiple velocity inlet BCs.

4.3 Multi-Part Mesh

Although the solid structure and internal fluid structure are discussed separately, they are meshed together. The framework offers the possibility to generate multipart meshes using multiple meshing algorithms. Thus allowing for coincident nodes to be automatically defined at the interface between two materials, eliminating often non-conforming interfaces with gaps or overlaps where two or more structures meet (Figure 7). This is useful and helps reduce simulation development time when there is a need to mesh multiple complex structures that are in contact, for example, when simulating a coupled heat transfer and fluid flow problem.

5. Export Model to COMSOL

Once the simulation-ready mesh and its BCs were defined, an .mph file is exported from ScanIP+FE and then imported into COMSOL (Figure 8).
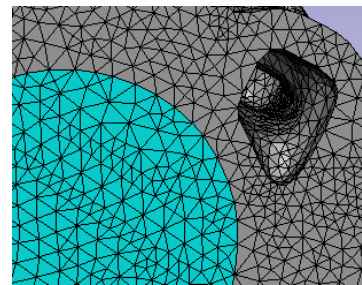

Figure 7. Illustration of coincident nodes defined at interface between solid (grey) and fluid (blue) structure.
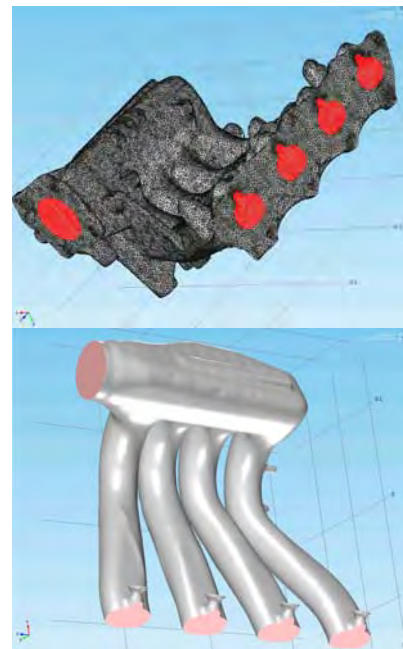

Figure 8. Within the COMSOL environment, the mesh of the solid (black) and fluid (red) structure in are shown (Top) and the CFD mesh of the fluid (grey) with the BCs highlighted (pink) are shown (Bottom).

COMSOL multiphysics software is applicable to many real-world mechanical applications in material analysis, part modeling, physics simulation, and performance evaluation. For fluid-structure interactions, the COMSOL CFD module can interact with its Mechanics module

for sophisticated physics interfaces simulations and studies. In reverse engineering it can be used to model part geometry, analyze part durability, test part crashworthy and safety. It has been used to improve product performance in sports equipment, automotive brakes, component seal, fasteners and piping fixtures. It has a unique capability to conduct analysis on geometric nonlinearity of mechanical components that might result from inherent material characteristics, buckling deformation, effects of loading mode or temperature.

6. Conclusion

The ability to automatically convert any 3D image dataset into high quality meshes is becoming the new modus operandi for reverse engineering. New tools for image-based modeling have been demonstrated, improving the ease of generating meshes for computational mechanics and opening up areas of research that would not be possible otherwise. Scanning modalities such as CT allow researchers to preserve and model the internal features of a structure while external laser scanning and probe methods do not.

The Simpleware software environment provides distinctive features and unique advantages allowing the reconstruction of a solid 3D model with fine precision of the scanned engine part, including its internal features. The utilization of COMSOL multiphysics software for further physics simulation and structure analysis also assures the functionality and performance of the reverse engineered part comparable to the original part.

7. References

1. Young, PG et al., An Efficient approach to converting three-dimensional image data into highly accurate computational models. *Phil. Trans. R. Soc. A,* **366**, 3155–31732 (2008).
2. Sethian, JA., *Level set methods and fast marching methods, evolving interfaces in computational geometry, fluid mechanics, computer vision, and materials science.* Cambridge Monograph on Applied and Computational Mathematics. Cambridge, UK: Cambridge University Press (1999).

3. Cebral JR. and Löhner R., From Medical Images to Anatomically Accurate Finite Element *Grids. Int. J. Num. Meth. Eng.*, **51**, 985-1008 (2001).