Simulation of Airflow in Building 661

Geometry Creation Software

A variety of AEC CAD applications exist to create drawings for new and existing construction projects. These applications vary in features, functionality and cost, but all support a subset of geometry export formats that may be the basis for 3D mesh generation tools targeted at Computational Fluid Dynamics (CFD) application input. A high level summary of some of the tools that were considered during the year 1 GPIC activity are described in the next section. The common file formats which the programs can import or export and listed. In addition, there is another set of formats that may be useful as a basis for creating 3D geometric mesh. These formats include Autodesk 3D StudioMAX binary format, **3DS,** Autodesk Maya binary format**, OBJ,** Autodesk Softimage **XSI,** Autodesk Kaydara **FBX,** Web-friendly **VRML and** Collada **DAE.**

CAD Tools

Google Sketchup Pro

Sketchup is a basic drawing package offering support for some commonly used geometric data formats typical of other popular drawing applications. The pro version of Sketchup, for which a fee is charged, is required. Sketchup Pro, allows export of 2D vector images in PDF and EPS format, allowing access from vector editing programs like Adobe Illustrator and Freehand. This is particularly useful for embedding in other documents where you require resolution independent images, but not necessarily useful as a basis for creating 3D geometric meshes for CFD

Autodesk AutoCAD

Autodesk AutoCAD formats DXF and DWG can be imported and exported by Google Sketchup Pro. This not only provides data flow between Sketchup and AutoCAD, but also other popular applications that also support these formats. Section definitions, layers, and scale are preserved during the import/export process.

Salome Open Cascade

This open source application exports BREP, IGES, STEP, SAT, STL, VTK formats. It is also part of a larger framework of applications that includes Code_Saturne, a popular CFD application used by the French nuclear reactor design industry to model heat transfer in nuclear reactors.

Cubit

Sandia National Laboratory developed cubit. It has a $10K per seat licensing fee for commercial users. There are complexities in the licensing terms that some

commercial legal departments may find challenging. It exports STEP, IGES, SAT, STL, and other formats.

Autodesk Revit

Revit is one of many design tools offered by Autodesk. Others include 3D StudioMAX, Maya, AutoCAD, and Softimage. These alternative applications are targeted at other industries that include Media & Entertainment, Automotive and Aerospace. Revit specifically targets AEC. Revit has a per seat cost of approximately $6500. It is a de facto industry standard and provides a full set of Building Information Modeling capabilities. It primarily exports RVT (native format), DXF, DWG (other Autodesk application formats), IFC (buildingSmart alliance) and IGES with optional export filter module.

Bentley Arechitecture/AECOSim Building Designer

Bentley offers a suite of AEC Design and Simulation Tools in addition to Bentley Architecture which is specifically targeted at architectural drawing for BIM. The cost of Bentley per seat is $??. Support of **DGN, DWG, DXF, PDF, STEP, IGES, STL**, and other major industry standards including import/export of IFC 2.x and gbXML.

3D Geometric Mesh Generation

There are a several open source and commercial tools available for the generation of 3D meshes to be used by CFD or structural mechanics applications. In addition to these tools, there are many proprietary tools in use for meshing, as this process is considered to be complex enough to benefit from customized algorithms that can provide a better mesh in terms of CFD performance and mathematical robustness.

Some of the publically available tools for meshing are highlighted in the next sections of this document.

OpenFOAM snappyHexMesh

OpenFOAM is an open source toolkit provided by Silicon Graphics (SGI) and can run in a distributed parallel environment using MPI. It is limited in the types of mesh cells and mesh structure it can generate. Its user interface is awkward and geometric formats supported are STL (stereo lithography) with only hexahedral mesh support.

Salome meshing component

Salome's meshing component is open source, but its limited meshing capability is not adequate for most problems. It imports BREP, IGES, STEP, and SAT formats

Cubit (from Sandia)

Application has a commercial licensing fee of 10K per seat and complex licensing terms. It has complete meshing capability but not parallel processing support implying that performance will not scale well as the mesh sizes or fidelity increase. Cubit imports STEP, IGES, SAT and STL.

Other commercial meshing software

ICEM-CFD from Ansys, which replaces Gambit, appears to be a powerful mesher.

CFD Workflows

CMU Sketchup Pro Plug-in

Initially we investigated using geometry generated in Google Sketchup Pro, exported as a .vrml file. The geometry was generated from a plug-in application for Sketchup developed by CMU. The applications run on a Microsoft Windows PC. One program inputs a VRML file and creates a .MESH and a .bStr file. The second program inputs these two files to create a .MSH Ansys Fluent mesh file.

Although, the Sketchup tools may be generally useful, it has some limitations for our specific purposes. First, most professional building retrofit or new construction projects will most likely rely on Autodesk Revit or Bentley Architecture applications to generate initial 3D CAD surface drawings. Sketchup may become popular over time, but that is not the current state of the industry.

The Sketchup plug-in does produce a Fluent mesh file, but it contains shapes that the OpenFOAM import filter could not process. This is a typical problem with import filters – they are never as robust as the native format. The Sketchup plug-in only produces hexahedral meshes at this time although it could be enhanced to include tetrahedral meshes. This is an important factor controlling mesh fidelity and quality near curved surfaces. Finally, It is not clear how to transfer boundary condition information into codes other than Fluent.

The limitations are certainly addressable, but not in plan at CMU at this time. In any case, this is a significant first attempt to provide an open source-meshing tool that integrated into Sketchup, which could be enhanced over time to provide a useful framework for 3D surface to 3D volumetric mesh generation for GPIC as well as Building Technology Program toolset.

Revit->Fluent conversion

In this scenario, we began with a Revit geometry file representing a drawing of building 661 at the Navy Yard. Revit, as previously noted, provides a variety of export filters including formats for other Autodesk products such as AutoCAD. We

Simulation of Airflow in Building 661

used Revit to generate the AutoCAD native DWG format. DWG format can be imported in AutoCAD and then exported to a SAT file by the AutoCAD export filters. SAT file can be imported into Ansys Gambit where the geometry can be simplified and refined before a mesh was built and exported as a Fluent .MSH file.

Using the Revit->Fluent workflow we have converted several building 661 meshes to an OpenFOAM format.

A key challenge is definition of boundary conditions. This information is not preserved from initial AEC CAD drawing. Using Fluent text files with boundary information encoded, we were able to modify the OpenFOAM SIMPLE test case to specify the correct boundary conditions for building 661, but automatic specification and conversion of boundary conditions is a critical step in the process that needs to be addressed going forward.

OpenFOAM Simulation

Several experiments were performed using the OpenFOAM CFD toolkit. OpenFOAM is attractive because it is free for both academic and commercial use, it is widely used for a wide range of CFD problems, it is parallelized for solving large problems on HPC machines, and it could be offered as part of a cloud computing environment. OpenFOAM provides the user with a class library of functions that can be used to develop domain specific applications. In this case, we used the simpleFoam solver to perform a simulation on our model of building 661. As previously noted, the mesh was created from the initial Revit model of building 661 and the mesh was converted to the OpenFOAM format using the fluentMeshToFoam utility which comes with OpenFOAM. The boundary conditions specification was done manually for our test as this remains one stage of the process that is not yet automated.

The OpenFOAM simulation was run on an Intel Linux machine, but given the low fidelity and resolution of the building 661 mesh, we did not require a lot of processing power to complete the simulation. The results are shown in the next 2 figures, which were produced by applying an OpenFOAM specific visualization tool called ParaFoam to the data results of the OpenFOAM simulation. This tool leverages the Kitware developed visualization application, Paraview and an OpenFOAM data reader to visualize the original building 661 mesh and map the pressure and velocity information produced during the simulation to the cells of the mesh to color intensities via a lookup table shading function that is part of Paraview application.

The OpenFOAM flow velocity components and magnitude are compared in Fig. 3 to a simulation of a similar problem carried out at Purdue using Fluent. There is agreement within a few percent except near the left boundary, where the Fluent result appears not to agree with the specified boundary conditions.
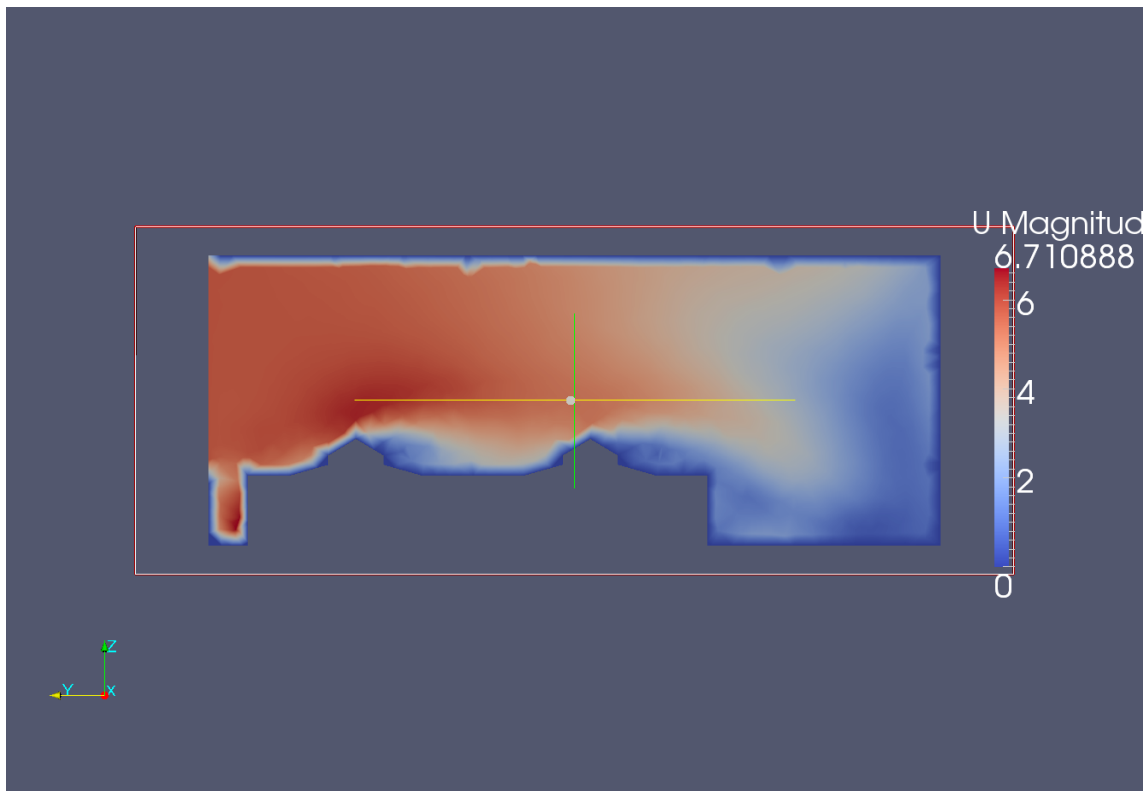
Simulation of Airflow in Building 661

Figure 1 Cross section of building 661 sampling airflow along the X axis at slice 30

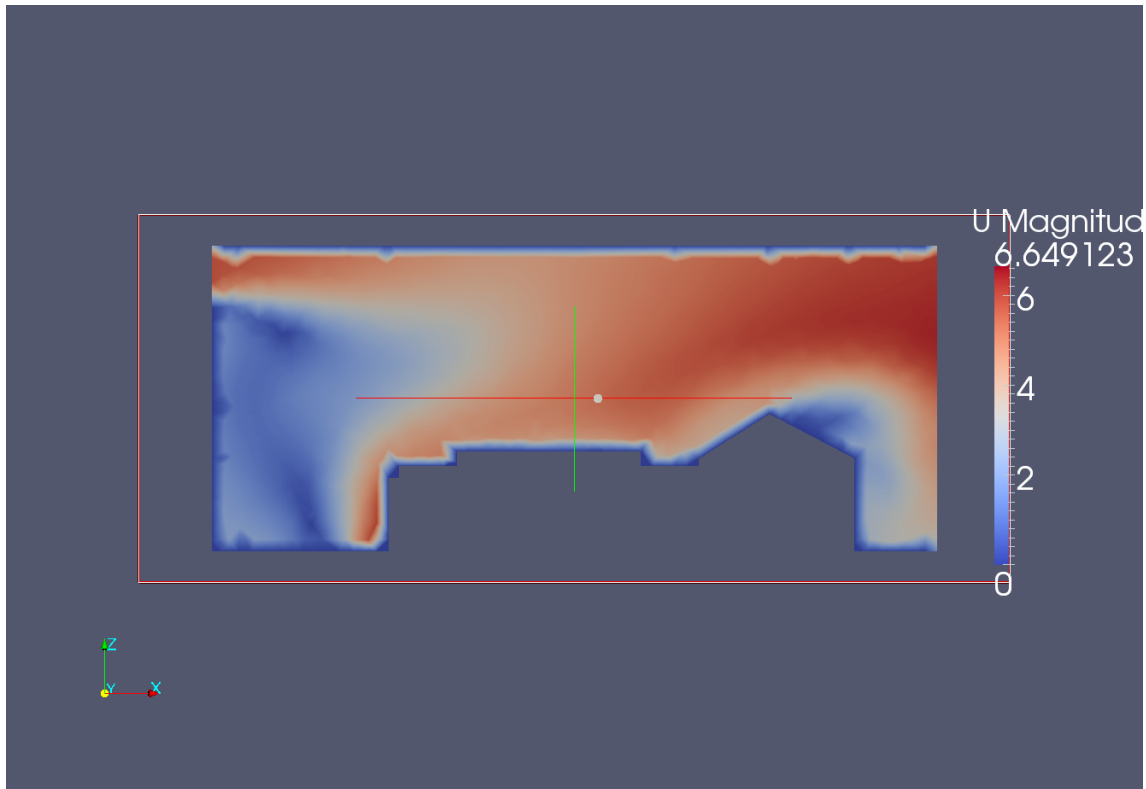# Simulation of Airflow in Building 661

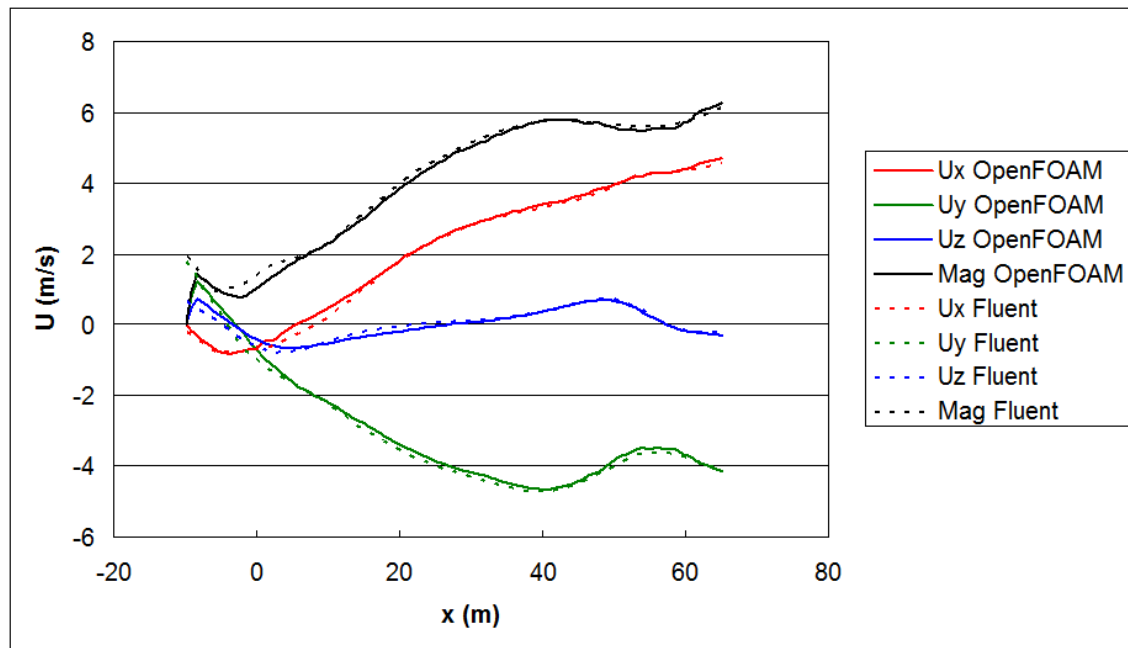**Figure 2 Cross section of building 661 sampling airflow along the Y axis at slice 20**

Figure 3 Comparison of OpenFOAM and Purdue Fluent simulation results for the Building 661 airflow along the line y=20 m, z=20m.