Requirements for the CAD to CEM process within ICE NITe

Henry Bucklow, TranscenData Europe Ltd.
`jhb@transcendata.com`

Simeon Earl, BAE Systems
`simeon.earl@baesystems.com`

Chris Jones, BAE Systems
`chris.c.jones@baesystems.com`

Geoff South, BAE Systems
`geoff.south@baesystems.com`

15th October 2014

ICE NITe aims to make a step change in the development of novel electromagnetic integration technologies—cable systems and connectors—in the aerospace industry. Key within this is the ability to perform computational electromagnetic simulations (CEM), and the main bottleneck in the CEM process is the connection between CAD geometry and CEM simulation. This document describes the requirements of the ICE NITe partners for an improved, automated, interface between CAD and CEM.

1 Introduction

Modern aircraft contain an ever-increasing array of complex, critical, electronic systems. Although the electronics themselves are increasingly sophisticated, approaches for integration and installation have changed little in the last 60 years. ICE NITe seeks to make a step change in the development of novel electromagnetic integration technologies, to reduce mass and volume of cable systems, whilst also lessening susceptibility to induced effects from lightning strikes and radiated threats. Key to the approach in ICE NITe is the ability to perform computational electromagnetic (CEM) simulations to evaluate the potential technologies.

In order to perform CEM simulations efficiently, ICE NITe seeks to improve, and automate, the interface between CAD modelling, and CEM. CAD models are typically produced to support packaging and manufacturing concerns, and are generally unsuitable for direct use in simulation. Instead, CEM analysts must spend significant amounts of time manipulating CAD models, to produce a geometry definition which is suitable for CEM simulation. Often, this process takes the majority of a CEM analyst's time, and is the bottleneck in the overall CEM process.

CADfix [1] is a tool for translation, repair and defeaturing of CAD models. It is used as a preprocessor for CEM analysis by a number of commercial vendors, and is BAE Systems' tool of choice for CEM preprocessing and meshing. CADfix is also used for repairing and defeaturing CAD models for other simulation disciplines, such as stress, thermal, and fluid dynamics, which share common requirements with CEM. This makes CADfix well placed as a tool for improving the CEM process. Within the ICE NITe project, TranscenData will enhance and extend CADfix to improve and automate the CAD to CEM process.

This document lays out the requirements stated by partners in ICE NITe. Section 2 describes some of the overall challenges posed by the CAD to CEM process, section 3 introduces some commonly used CAD and simulation terms, section 4 describes the existing CEM workflows used by the ICE NITe partners, and section 5 goes into further detail about the requirements imposed by these workflows.

2 The challenge

The partners in ICE NITe commonly need to run CEM analyses on complex models. For example, a complete aircraft CAD model may contain as many as 30,000 separate bodies, and future aircraft will only be more complex. Regardless of the technology used for CEM simulation, a degree of model preparation, and defeaturing, is necessary in order to perform simulations on models of this size.

It's also clear that the CAD to CEM process for models of this size needs to be largely automated. The CEM analyst must only be involved in manual decisions where necessary, and the decision making process must be streamlined, otherwise they will be swamped.

Automated processes must also build trust with their users, by reporting and summarising the results of the automated processes in an understandable way. Without confidence, automation is largely worthless, as the automatic tools will never be used.

3 Terminology

This document will use terminology common in the CAD modelling world. A CAD assembly is a collection of *bodies* (figure 1), each one representing a single solid object. Each body is defined by describing its boundary, as a collection of *faces*, which join together at *edges* (figure 2). The geometry of each face is described by a smooth mathematical function. Typically, each edge is connected to exactly two faces; such a body is described as *manifold*.

Often, CAD geometry is simplified by converting it to a *facetted* representation (figure 3). This approximates each body by a collection of connected triangles. Such facettings are sometimes easier to work with, and are used in several standard file formats, such as STL.

Before geometry can be used for simulation, it usually needs to be *meshed*. This takes the boundary representation used by the CAD model, and converts

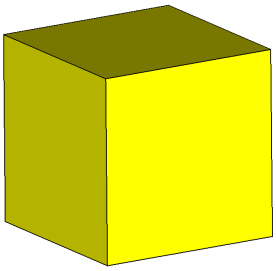


Figure 1: A simple cube body.

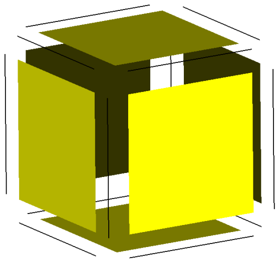


Figure 2: The cube, broken down into constituent faces and edges.

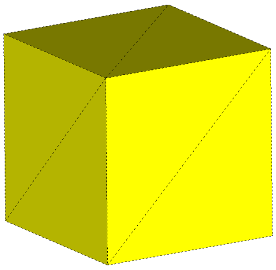


Figure 3: The cube, as a facetted model.

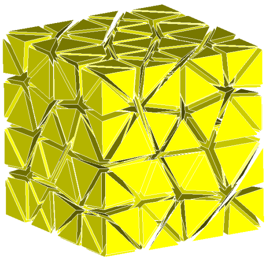


Figure 4: The cube, meshed with tetrahedral cells.

it into a representation of the volume. A collection of connected, small, simple, geometric shapes, known as *cells* are used to represent the model; typically either cubes, or tetrahedra, are used in CEM (figure 4). The corners of the cells are referred to as *nodes*.

4 Workflows

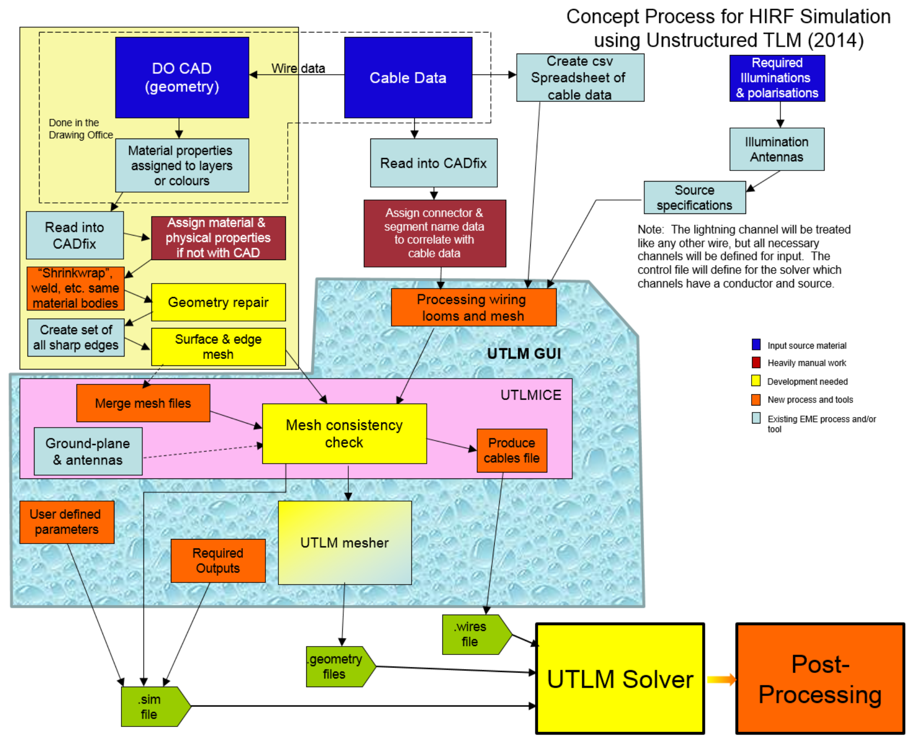
As there are several partners performing CEM within ICE NITe, there is no single fixed workflow which will be followed in the project. Instead, in this section, the partner workflows will be documented, so that their requirements can be understood.

Common to these workflows is the idea that CAD models are not directly suitable for CEM simulation; they must first be converted into a form which expresses the correct electrical connectivity between parts, at the appropriate level of detail, before they can be meshed and finally simulated.

4.1 BAE Systems and Nottingham University

BAE Systems use CEM for a variety of analyses on military air platforms, components, and technologies. Commonly, aircraft models are examined for their response to lightning strikes, and high-frequency radiated threats, and this is an important part of the development and certification process.

BAE Systems have an established method for CEM, based on the parallel transmission line method [2] (PTLM). This uses a Cartesian mesh, where the geometry is broken into small, uniform, axis aligned cubes, and BAE Systems



Note: The lightning channel will be treated like any other wire, but all necessary channels will be defined for input. The control file will define for the solver which channels have a conductor and source.

Figure 5: Diagram of BAE Systems workflow using the Nottingham University UTLM mesher and solver.

are finding significant limitations with the level of detail they can achieve on large models, as the same cell size must be used across the whole model.

In contrast, the unstructured transmission line method [3] (UTLM) being developed by Nottingham University uses tetrahedral cells, which can be varied in size across the model, allowing high detail levels to be used where appropriate, and lower levels of detail elsewhere. This allows a higher fidelity of model to be processed for a similar computational budget.

BAE Systems have developed a new process, using UTLM, which they are planning to advance during ICE NITe (figure 5). As such, this is the process that is documented here.

4.1.1 Model assembly at BAE Systems

Most of BAE Systems CAD is produced in Catia and is primarily used for the manufacture of parts. However, the engineering side of the business is increasingly using the design data to predict and optimise the performance of various systems and sub-systems, and the electromagnetics discipline uses CAD as an input to a number of aircraft analyses.

CAD models represent each part of an assembly as an individual solid body, held in close proximity to its neighbours. There is no mathematical connection between neighbouring bodies[1]. Before such an assembly can be used for CEM,

---

[1] Aerospace engineers joke that an aircraft is merely thousands of parts flying in close

it must be converted into a *cellular model* [4] [5], where boundaries between adjoining parts are explicitly shared between them, and new bodies are created to represent air gaps, welds, and fillers between parts (§5.1).

The use of cellular models is necessary to ensure that correct electrical connectivity is simulated. It also offers opportunities for simplifying the CEM model, as connected bodies with identical materials can then be joined into a single body. This reduces the constraints on the UTLM mesh, allowing a faster, more robust meshing process, and larger cell sizes.

Once a cellular model has been created, the level of detail in the model needs to be reduced, by a process of *defeaturing* (§5.2). Excessive detail causes a number of issues:

- Difficulties generating a mesh,
- Slow meshing process,
- Large cell count, and
- Small cell size.

Difficulties in the meshing process can mean it is impossible, or takes a prohibitive amount of time, to produce a simulation model at all. Large cell counts, and small cell sizes, can lead to simulations which need more computational resources than are available. Since the simulations being run by BAE Systems are often on the edge of what is possible, defeaturing can be very important.

Another key piece of information for CEM simulation is the location and definition of the wiring within the model. BAE Systems use the CAD definition to provide the shape of the channel through which the wires must pass, whilst the actual composition of each wire bundle, and the electrical properties, are stored in an external database. This data is combined (see figure 6) to produce a *wires file* which can be passed to the UTLM solver.

Non-CAD information also needs to be attached to the CEM model. Each body requires a material tag, and faces may also have tags for surface properties, to represent surface layer properties, such as a coating. As, in general, the UTLM mesher does not respect face boundaries, it is necessary to define edges that the mesher must respect, to preserve sharp, geometry, and to generally aid the mesher. These edges are are tagged as sharp lines. Wires must also be tagged – each wire requires at least two, and up to four, tags, with each wire end having a separate tag.

All these tags are generally names, be it a connector name for a wire termination or a material name for a body. There is one requirement on the names, which is that a tag name can only be used to convey one type of property, for example a body material tag cannot be used as a wire name. The tags are treated as names, and pass through the UTLM mesher to the solver without being interpreted. This allows material properties to be altered without regenerating the mesh. The solver can then look up the tags in the simulation control and/or wire definition files to access the actual electromagnetic properties.

A facetted representation is generated from each body in the cellular model, and this is used to represent the geometry to the Nottingham University UTLM

---

formation (the closeness being key), but this is exactly how a CAD system represents the geometry.

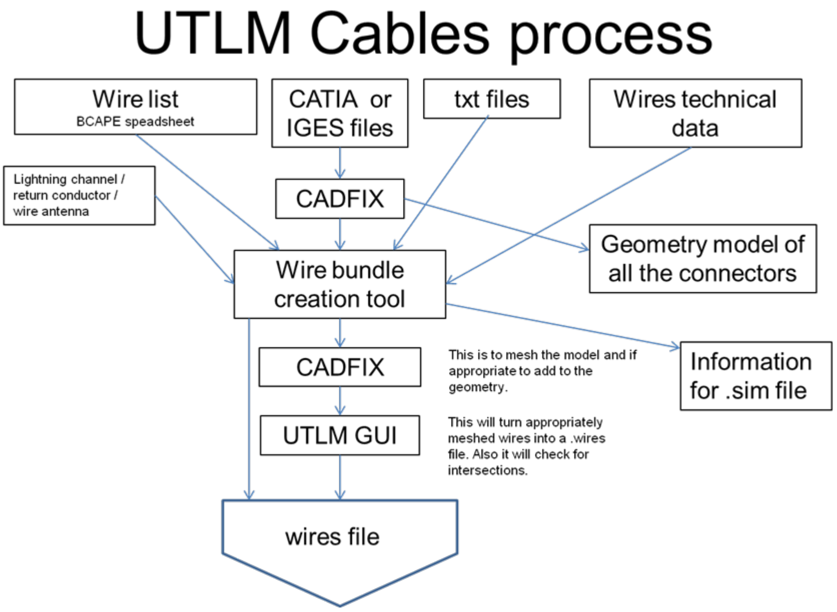


Figure 6: Producing wire definitions for the UTLM process at BAE Systems.

mesher and solver. BAE Systems have a tool which checks, using the CADfix API, that a model is valid before sending it to the UTLM mesher:

- Each body must have material properties
- Each surface must have the necessary properties
- All wires must have radius and conductivity information
- Facettings must be watertight, without self-intersection
- Wires must not intersect bodies
- Wires must not intersect each other
- Wire radii must not interfere with each other
- Wires must have a termination defined; earthed definitions must contact a surface
- Junction definitions must contact at least one other wire

Finally, the facetting, together with material and interface information, is sent to the Nottingham University UTLM mesher in a custom file format – *Simple Geometry Format* (SGF). The non-geometric information—the simulation control and wire definition information—are passed to the UTLM solver in `.sim` and `.wire` files respectively.

4.1.2 The Nottingham University mesher and solver

The George Green Institute at Nottingham University has a considerable international reputation in the field of CEM. For the past few years, they have been developing their UTLM mesher and solver, which provide a more efficient way to perform large scale CEM simulations.

Ultimately, the Nottingham University UTLM solver requires a Delaunay tetrahedral mesh (where the circumsphere generated by the vertices of each tetrahedron contains no other vertices), together with boundary conditions, solver parameters, and wire definitions. This mesh may include degenerate tetrahedral cells, and a clustering algorithm allows small cells to be handled without forcing a small timestep. The Delaunay criterion must be obeyed strictly, which is not something most commercial meshing tools can provide; they tend to produce mostly Delaunay cells, but will often disregard the Delaunay criterion, in order to mesh more robustly.

Because of the strict meshing conditions, the UTLM solver is typically used with the Nottingham University mesher. This accepts a range of formats:

- IGES files (with some capability to repair poor geometry).
- SGF files - a triangulation representing the geometry of the object to be meshed. These meshes are then parameterised and re-meshed.
- A *Planar Facetted Model*, a Nottingham University format for defining planar polyhedral models.

A set of global meshing controls are also provided, with some simple local controls to mesh certain areas with an increased density.

When SGF files are used as input—as is the case within BAE Systems—the mesher is freed from respecting the topology of the CAD model, which improves the success of the meshing process. The density of the triangulation in the SGF file is still important, however, as excess triangles make the parameterisation process slow, which subsequently holds up the meshing process.

Delaunay meshing of some configurations can be difficult – sharp angles, for example, can cause difficulties, as can interactions in narrow areas. Typically, these are resolved by defeaturing the model prior to meshing (§5.2).

Although there is no requirement for the mesh to have a particular structure, the solver does benefit from the use of mid-surfaces, or prismatic cells (where each node is directly opposite a partner on the far wall) to represent thin walls (see figures 7–12). These can then be used with a specific electrical model to improve accuracy and efficiency when modelling thin walls (§5.4).

The mesh does not have to respect the wires in the CEM model. Instead, the wire definitions are passed separately to the solver, where they are used to modify the electrical properties of each tetrahedron which they pass through.

The tetrahedral mesh, boundary conditions, solver parameters, and wire definitions are finally passed to the solver. This produces a set of observations, in the form of raw numbers, results surfaces and graphs.

4.2 MIRA workflow

MIRA use CEM for a wide range of simulations relating to automotive design and development, from studies of electromagnetic susceptibility of vehicles, to

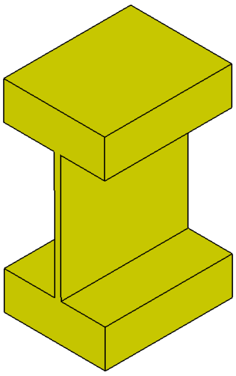


Figure 7: An example model with a thin-walled structure.

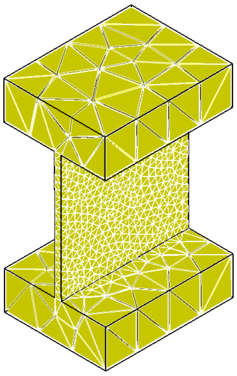


Figure 8: A typical tetrahedral mesh for this structure. Note that small tetrahedra are used to mesh the thin wall – this is typically needed, to maintain sufficient mesh quality.

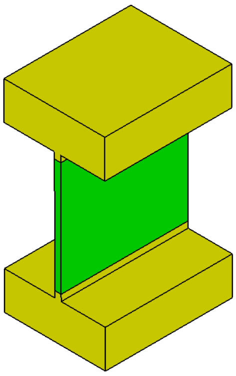


Figure 9: An example model, with the thin region divided into a separate body.

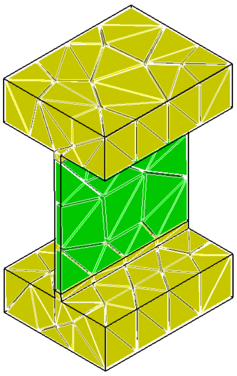


Figure 10: A typical mesh for this structure, using prismatic cells in the thin region. Note that far fewer cells are needed.

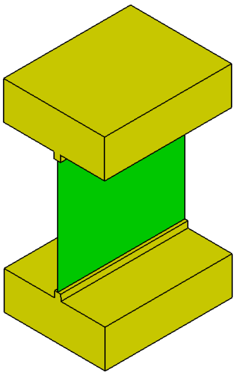


Figure 11: An example model, with the thin region reduced to a midsurface.

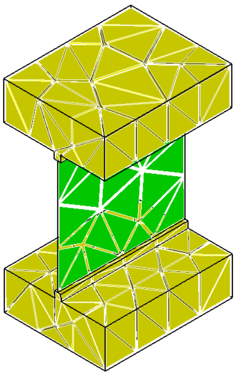


Figure 12: A typical mesh for this structure, using triangular cells on the thin region. Note that far fewer cells are needed.

predicting the electromagnetic energy absorbed by passengers. As an independent research organisation, they receive CAD geometry from many different customers, with varying complexity. Commonly, though, their models will contain complex surfacing, such as vehicle bodywork, which can be time consuming to prepare for CEM.

MIRA use a variety of commercial CEM codes, including:

- CST MicroStripes, a TLM code (like PTLM),
- FEKO, used for its boundary element code,
- Flux3D, a finite element code, and
- SemCAD, a code specifically for modelling the human body.

However, the workflow is broadly similar for all of these codes. Initially, the CAD model is filtered, within the CAD system, to remove parts below the required level of detail. Subsequently, parts with a similar material are joined together, and defeatured. Increasingly, this is now done using a shrinkwrapping approach (§5.3), where a facetted representation of the united parts is produced at a fixed resolution, automatically suppressing gaps and small features below this size.

Facetted models (as STL files) are used as the geometry input for all the CEM solvers used at MIRA, which fits well with the shrinkwrapping approach. Parts made from different materials are facetted separately, producing multiple STL files, which are then united again within the solver environment.

5 Requirements

From these workflows, and discussions with the ICE NITe partners, a set of requirements has been distilled, which will be described in this section.

5.1 Establishing connectivity

As described in section 4.1.1, BAE Systems need to convert their CAD models into cellular models, for their UTLM workflow. Neighbouring bodies, which are geometrically coincident, need need to be converted into bodies which share faces, by the process of *imprinting* (§5.1.1). Gaps between bodies need to be closed, by introducing new bodies representing welds, fillers, or air volumes, by a process of *welding* (§5.1.2). In the resulting model, all the bodies that are meant to touch their neighbour should do so, with no body intersecting another body – the geometry needs to be consistent and connected.

As a full aircraft model contains a large number of bodies, automation is essential to making this problem tractable. This must be accompanied by suitable reporting, and guided manual processing where necessary, to build trust in the process.

5.1.1 Imprinting

Two bodies, which have one or more faces that are geometrically coincident (to within a given tolerance), can be *imprinted*. This scratches the edges of

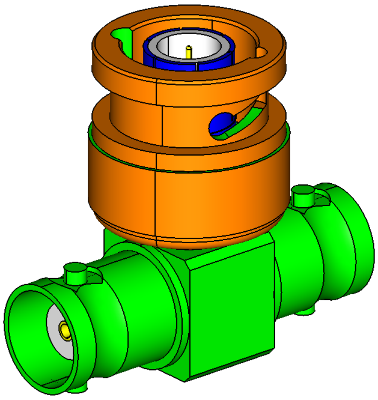


Figure 13: An example model for imprinting.

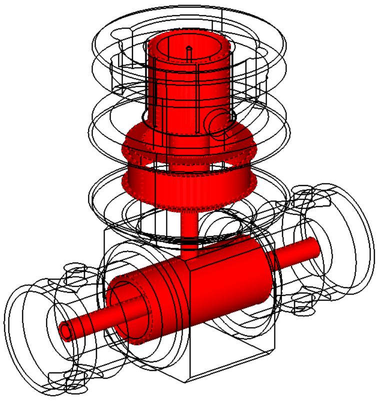


Figure 14: The imprinted surfaces for this example model.

the coincident faces into both bodies, and creates a new face definition which is shared between the bodies. See figures 13 and 14 for an example of how imprinting can be used on a simple model of a BNC connector.

Imprinting is a very common operation; it's expected that nearly all solids in a CEM model for BAE Systems will require one or more imprinting operation. This means that the imprinting process needs to be as automatic and robust as possible.

Given the large number of imprinting operations, it's important that any deviation introduced by the imprinting process is controlled and reported:

- The permitted deviation should be set directly by the CEM analyst, as they can relate the deviation directly to the CEM discretisation error - changes to the model which are significantly below discretisation error should be acceptable.
- For engineers to have confidence in the tool, it must also provide reassurance that the model has not been altered significantly. For example, it could provide a report covering the maximum deviation across all bodies, together with a breakdown of the maximum deviation in each body.

Occasionally, it will not be possible to imprint bodies without altering the model by more than the permitted deviation, or without damaging the model topology. In these cases, the imprinting tool should display the relevant bodies after the automatic process has completed, to allow the CEM analyst to make any necessary changes manually.

5.1.2 Welding

Commonly, two bodies, which are attached to one another during manufacture, will have a small gap between their geometrical definitions. This can be an error in the CAD model, or the deliberate omission of a shim or adhesive from the model.

To connect these bodies, a small shim body needs to be produced which will connect the two; we term this *welding*. See figure 15 for an example weld.

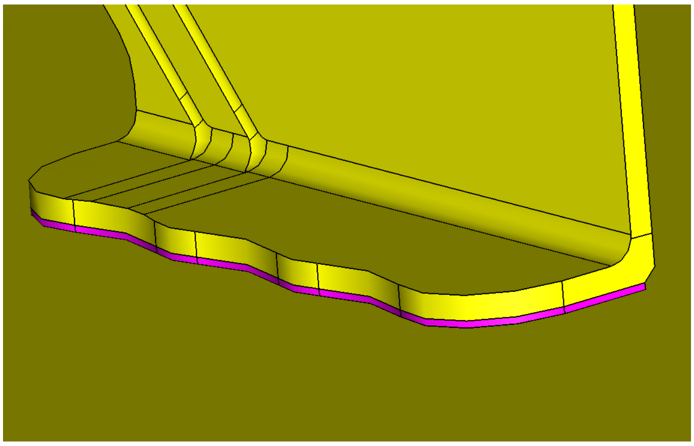


Figure 15: An example weld (in pink) between a wing skin and a bracket on a BAE Systems model.

Rules, or engineering judgement, will need to be applied to choose the material for this new body, depending on the bodies being connected.

The welding process could be required to run on a significant number of bodies. It will be necessary to detect bodies suitable for welding automatically, as manually searching for air gaps in a whole aircraft assembly will take a prohibitive amount of time. It may be possible for welds to be automatically constructed between bodies of certain materials, under controlled geometric conditions, but more typically decisions over whether to weld bodies, and how to do it, require an engineer with knowledge of how the product will actually be constructed.

This makes it important to have a user interface which makes it quick for the CEM analyst to examine each potential weld, and make a decision about how it should be processed.

It's important that the volume of added material is controlled and recorded, to avoid making the resulting model non-physical. Warnings should be raised if a weld is used to bridge a gap which is significantly below the target cell size, as the mesh will be affected.

Another possible mistake which could be caught by the welding tool is the accidental introduction of intersecting bodies. For example, a failure to include a washer in the bodies being welded would introduce an intersection, as in figure 16. These problems can be avoided by helping to automate the weld detection process, and by running intersection checks after the weld is complete.

A useful variant of the welding process would be to detect specific configurations of bodies where known CEM models exist. For example, butt joints (figure 17), and overlap joints (18), have specific models in Microstripes, which MIRA would be able to exploit.

5.1.3 Body joining

To reduce model complexity, it may be necessary to join bodies of similar materials, removing their interface surfaces. This is the main strategy within BAE Systems for improving mesh quality within thin components – these are often

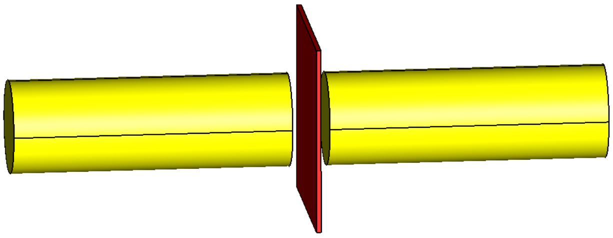


Figure 16: Creating a weld between the two yellow bodies would introduce an intersection between the weld, and the red body, forming an invalid model.

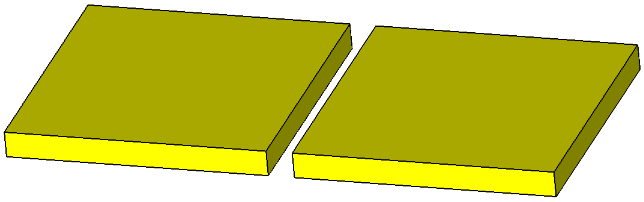


Figure 17: An example butt joint between two plates. The gap between the two plates must be measured as an input to the Microstripes model.

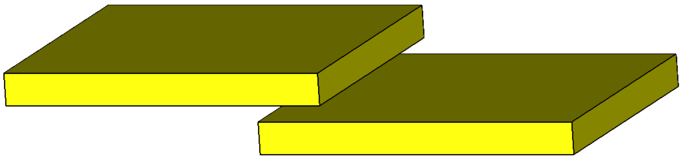


Figure 18: An example overlap joint between two plates. The length of the overlap must be measured as an input to the Microstripes model.

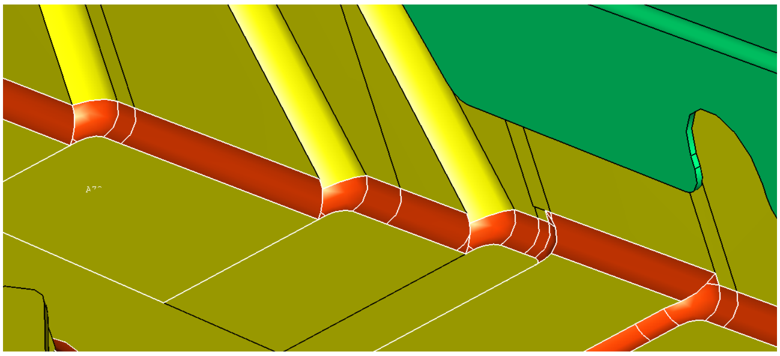


Figure 19: An example fillet (in orange) on a BAE Systems geometry.

combined with neighbouring components of similar materials to increase the thickness of the overall part.

Once bodies are correctly imprinted (§5.1.1), joining the bodies becomes a trivial operation. It may also be necessary to join neighbouring faces within the joined body, to remove surface meshing constraints (§5.2.4).

5.2 Defeaturing

Once a cellular model has been created, the level of detail in the model needs to be reduced. For many CEM methods, the overall cost of the CEM simulation is inversely proportional to the size of the smallest cell in the CEM mesh, which makes control of cell sizes in the CEM mesh critical to the computation cost of the simulation. Although the Nottingham University UTLM solver implements techniques to mitigate this issue, excessively small cells still impose a computational cost, and geometric features such as sharp angles, and complex interacting features, can make it difficult to produce the required Delaunay mesh.

5.2.1 Automatic fillet removal

Fillets are a common model feature, which introduce specific problems for meshing and simulation tools. See figure 19 for an example fillet on a BAE Systems CAD model. The tight curvature, narrow face width, and tendency to contain fragmented faces, all combine to drive cell sizes down, and CEM solution times up. Fillets also often contain faces with sharp angles, which can make Delaunay meshing fragile.

Although fillets can be removed using a general feature removal tool (see §5.2.3), this is not always a successful approach. When fillets interact with each other, and with other features, it can be very difficult to isolate them as a simple feature (figure 20). Instead, an approach which directly exploits the structure of the fillet is required.

As fillets are so common, it's important that their detection and removal can be automated as much as possible. Controls should be available to cover the size of fillets which should be removed, and the removal process should be safe and robust. If necessary, reports could be generated describing the change in volume and surface area of the de-filleted parts.

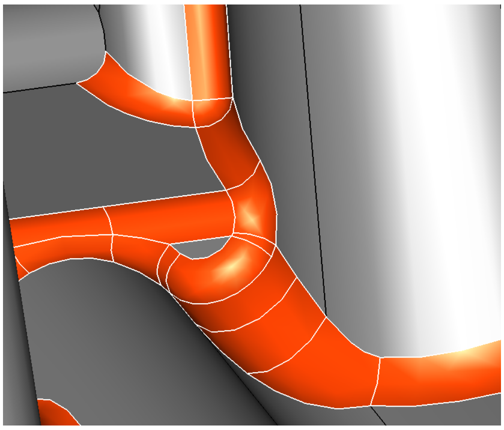


Figure 20: Interacting fillets, which are difficult to remove as a simple feature.

5.2.2 Steps

Thin steps are a very common feature of carbon fibre composite structures (figure 21). Whenever the number of carbon fibre layers changes through the structure, a step is introduced. These steps are often below the resolution of the CEM analysis, and need to be suppressed, to avoid introducing unnecessary cells.

This process will need to be manually guided, as the replacement of these steps will require engineering judgement. It's anticipated that once the step faces have been selected, a tool would then be able to replace them with a simpler and smoother piece of geometry. If necessary, the tool could provide feedback regarding the changes in volume and surface area as a result of the replacement.

5.2.3 Feature suppression

Within a CEM simulation, aluminium components can often be altered significantly without concern, as it's an excellent electrical conductor. General defeaturing tools, for deleting sections of a model (for example, see figures 22 and 23), and healing the resulting gap, are needed to help the engineer make these sorts of large scale changes.

This tool is expected to be used directly by the CEM analyst, rather than being automated. The tool should still be robust, and it must either remove the selected feature cleanly, or report that it cannot remove the feature - it must not damage the model.

5.2.4 Face joining

Excessive model topology (see figure 24 for an example) results in longer processing times for subsequent validation and meshing steps, as small and narrow faces force small triangles into the facetting of the model. These faces need to be removed, by joining them to their neighbours, creating new NURBS surfaces to represent the combined geometry.

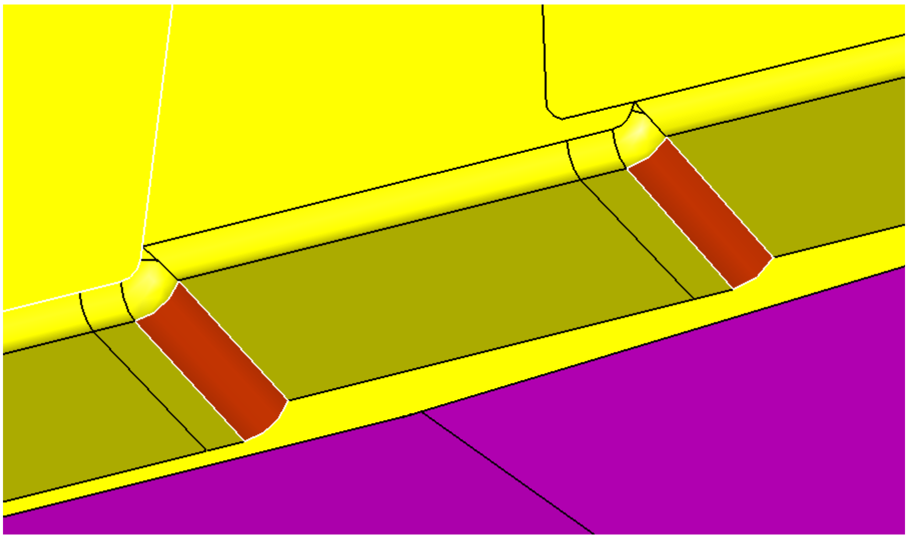


Figure 21: An example narrow step (with the step faces highlighted in orange) on a BAE Systems carbon fibre composite geometry.

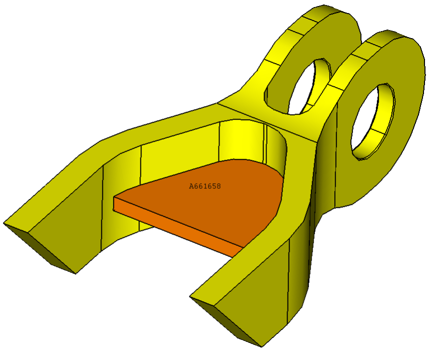


Figure 22: An example stiffening structure (in orange), in a BAE Systems geometry, which could potentially be removed for CEM.

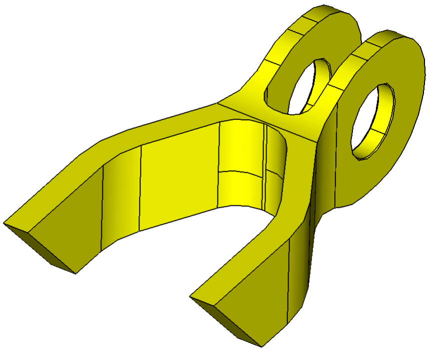


Figure 23: The model with the stiffening structure removed.

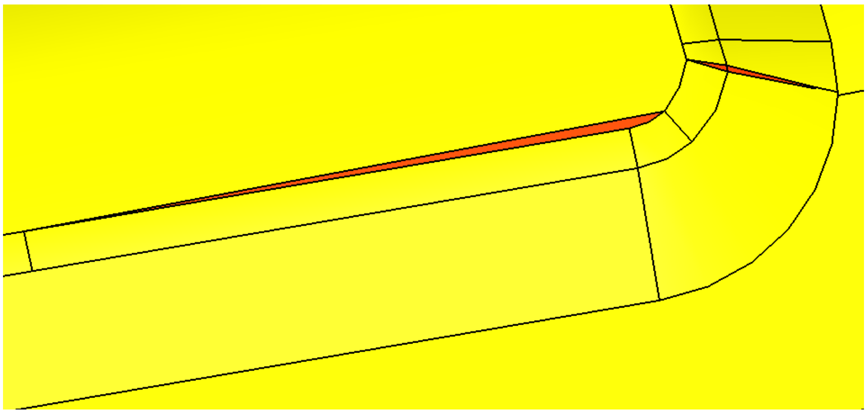


Figure 24: Excess face topology on a BAE Systems example geometry, with small faces highlighted in orange. This can be difficult to mesh, or lead to dense facets in a facetted representation, slowing down subsequent processing.

5.3 Shrinkwrapping

Shrinkwrapping is a technique for generating a facetted representation of a CAD model, with a fixed resolution, suppressing details, holes, and gaps below this resolution. It's intended as a "one click" method, where the CEM analyst simply sets some basic parameters, and the remainder of the algorithm is automatic. As such, it can replace the techniques already discussed for establishing connectivity and defeaturing. Shrinkwrapping is used extensively at MIRA, as their preferred method of preparing geometry for CEM. See figures 25 and 26 for an example shrinkwrap, on a geometry relevant to ICE NITe.

Shrinkwrapping is not suitable for all geometries; for high-fidelity analysis, the NURBS-based defeaturing techniques previously described in this document provide a greater level of control. It is therefore important that, within a CAD assembly, some parts can be shrinkwrapped, and others processed using NURBS-based techniques, with the two kinds of defeatured geometry interfacing correctly.

All properties stored on the CAD geometry - typically electrical and interface properties - need to be retained by the shrinkwrap, otherwise the CEM analysis will not be able to proceed.

It's also important that individual CAD edges can be preserved in the shrinkwrap, with a chain of facet edges identified as representing the edge, as boundary conditions may need to be applied to these edges.

The raw output from a shrinkwrapping algorithm is often a dense, poor quality triangulation, which can be difficult, and expensive, to use downstream. The shrinkwrap facettings which are generated in ICE NITe should be smoothed and re-meshed as necessary to produce a facetting with appropriately sized triangles of reasonable quality.

5.4 Thin regions

Thin structures pose specific difficulties for UTLM meshing, as they do for many other simulation methods based on tetrahedral meshes. In order to produce an

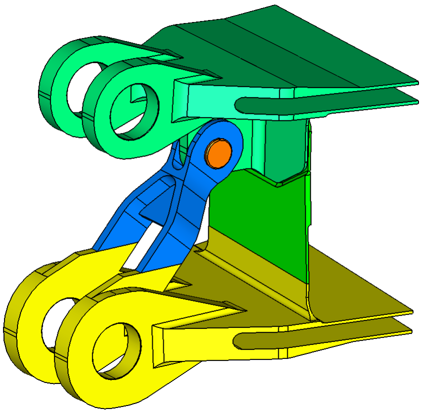


Figure 25: An assembly of bodies in contact, in a BAE Systems example geometry. The bodies have been randomly coloured, to help distinguish them.

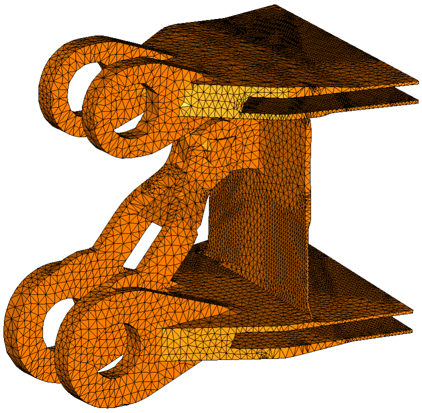


Figure 26: The bodies, after shrinkwrapping. A single skin of triangular facets has been created, around the outside of the bodies. The boundaries between them have been ignored.

acceptable quality solution on the thin region, the tetrahedra must be approximately isotropic - with all edges approximately the same size (figure 7). As a result, a large number of cells are needed to mesh thin regions, imposing a large computational cost. Although UTLM uses cell clustering methods to mitigate this cost, thin regions remain expensive to simulate. Thin regions are extremely common in aircraft and automotive models – see figure 27 for an example.

There are, however, techniques within UTLM which treat thin regions specially, and require a specialised mesh in order to do so. Either the thin region must be reduced to its mid-surface (figure 11), and meshed using triangles (figure 12), or the thin region must have a mesh with directly corresponding nodes on each side (a *prismatic* mesh, see figure 10). A special model is then used for each triangle or prism, to model its electromagnetic response, which does not couple the required width of the triangle or prism to the thickness of the material.

An automated method is needed for discovering thin regions, and either reducing them to their mid-surface, or meshing them using a prismatic method. Promising algorithms for both of these methods have been proposed, based on the *3D medial object* (figure 28), which is a dimensionally reduced, skeletal, representation of a body, and is extremely useful for analysing thickness, and generating mid-surfaces. It is also important that mid-surfaces, and prismatic bodies, are coupled correctly to their neighbouring bodies, and other thin regions. In the case of coupling between connected, overlapping thin regions, this means imposing matched meshes between two mid-surfaces, so that the nodes can be marked as electrically connected in the solution.

Thin components are often manufactured from carbon fibre composite materials. The production techniques result in the component containing multiple layers, with the carbon fibres running in a different alignment in each layer. These differences produce varying, anisotropic, electrical properties in

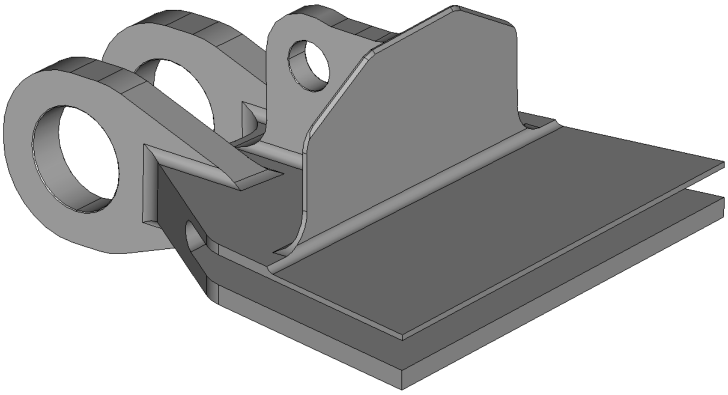


Figure 27: An example body with thin regions, from a BAE Systems CAD model.

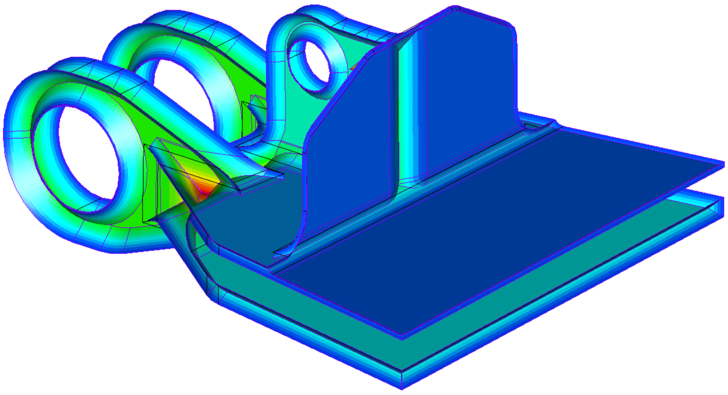


Figure 28: The 3D medial object of an example body with thin regions, from a BAE Systems CAD model. The 3D medial object is a dimensionally reduced, skeletal, representation of an object, and is a precursor to a mid-surface or prism-meshed version. The colours indicate the object thickness.

each layer, and as a result it can be desirable to model these layers as separate bodies (see §5.8).

The CAD model only defines the outer envelope of a composite component, rather than the individual layers. As a result, a method is needed to divide a thin body into a series of bodies estimating the layers of carbon fibre used in the manufactured part. The medial object provides key thickness and shape information for performing this operation, and it is anticipated that such a tool would use a derivative of the medial object to perform a series of *shelling* operations, to generate suitable layers. As such an analysis would be very high resolution, this tool would only be expected to be employed on small numbers of simple bodies, rather than complex aircraft assemblies.

5.5 Automation

The CEM processes developed in ICE NITe need to extend to large CAD models, with tens of thousands of bodies, making automation key. Although the methods described in this document vary as to how much input is needed from the CEM analyst, some automation is always possible. It's important that the automated process is:

- conservative about altering the model, with control over how much change is allowed, and
- instrumented, providing information about how much the model has been altered.

When automation is not possible, it's important that the CEM analyst be guided through the possibilities for fixing each model problem in an efficient way, to help them make progress as quickly as possible.

It's also extremely important that the metadata associated with the CAD model—materials, interface conditions, etc—are not lost during processing. Every tool developed within ICE NITe needs to ensure that metadata is preserved, otherwise the resulting model will not be useful for CEM.

5.6 Marking of CAD with non-CAD info

As described in §4.1.1, bodies, faces, edges, and wires must carry tags, describing their physical properties, and providing controls for downstream processing. Some tags—for example, the "sharp edges" tag—should be interpreted by the mesher, but most should be ignored (and passed on), and their definition looked up by the solver from the solver control file, or wire definion file.

It is expected that parts of this project will require additional tag types, particularly due to anisotropic material properties.

5.7 Cable route and bundle data

As ICE NITe is particularly concerned with structured cable systems, and their performance relative to existing practise, it will be important to accurately model cables within a bundle. This will require the generation of explicit cable bundle geometries from the enclosing tube definition, using the bundle cross section information to position individual wires. A method for modelling junctions and connections between these bundles must also be chosen.

Longer term, the project should influence current design practise, to allow more accurate CEM simulation of future aircraft, by providing explicit positions of cables in the design.

5.8 Handling anisotropy

Most modern aircraft contain composite parts. The composites used tend to have highly anisotropic material properties due to their design and manufacture. An example of such a material is *carbon fibre composite* (CFC). Depending on the fidelity of the modelling being undertaken, a composite component may be considered as a single body or may be required to be broken down into layers, or plies in the case of CFC. This layering does not exist in the CAD design but as an instruction to manufacture. This means that the layering needs to be regenerated, and added to the model, by breaking down the original body into a separate body for each layer.

Whether a single body is used, or a body for each layer, the anisotropy must be defined and oriented in the geometry, and communicated to the solver. The precise mechanism will be determined during the project, but the likelihood is that at least one vector will need to be passed from the geometry to the solver for each anisotropic body, to indicate the direction of the material.

5.9 Configuration control

It is mandated within the BAE Systems Operational Framework that all software, either produced or procured to provide through life Engineering Capability shall establish and maintain a requirements baseline. Within BAE Systems, output from ICE NITe will be managed using existing BAE Systems policies and procedures. In particular, the following will be used as the primary basis:

- Engineering Policy [Operational Framework, 759/OF/035]
- Configuration Management Policy [MASED-0028]
- Software Engineering Policy [MASED-0083]

These generic documents are designed to be used on all projects and therefore their entire content may not be applicable to ICE NITe. Where appropriate, tailoring will be used to ensure only relevant clauses are specified.

External to BAE Systems, the output will be managed according to individual company procedures.

6 Conclusion

This document has presented the CEM workflows of the ICE NITe partners, and described their requirements for an improved, automated, interface between CAD and CEM. This improved process will alleviate a major bottleneck in the CEM simulation process, and form the foundation for the project goal of making a step change in the development of electromagnetic integration technologies in the aerospace industry.

References

[1] TranscenData Europe Ltd.. CADfix 9.0. `http://www.transcendata.com/products/cadfix`

[2] Johns, P.B., Beurle R.L., Numerical Solution of two-dimensional scattering problems using transmission-line matrix, Proceedings of the IEEE, vol 118, no. 9, pp1203-1208, 1971.

[3] Sewell, P, Wykes, J.G., Benson, T.M., Christopoulos, C., Thomas, D.W.P., Vukovic, A., Transmission-line modeling using unstructured triangular meshes.

[4] R. Bidarra, K. J. de Kraker and W. F. Bronsvoort, Representation and management of feature information in a cellular model, Computer Aided Design, vol. 30, no. 4, pp. 301-313, 1998.

[5] R. Bidarra, J. Madeira, W. J. Neels and W. F. Bronsvoort, Efficiency of boundary evaluation for a cellular model, Computer Aided Design, vol. 37, no. 12, pp. 1266-1284, 2005.