A Process to Easily Create `.obj` Mesh Files for **42** from Your Existing CAD Files

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1 Purpose of This Document

This document serves to outline a viable process for converting existing CAD models from various programs to Wavefront Object `.obj` and Material Library `.mtl` files. This is the desired file input format for graphics and surface force calculations in the simulation program **42**, and may have uses outside this single workflow. This document assumes minimal familiarity with **42**, and may be skimmed by more experienced users. This version presents a purely open source workflow, though commercial CAD softwares may also aid the reader.

2 Things You Will Need

To do this, you will need the following software:

- FreeCAD version 0.16
  `https://sourceforge.net/projects/free-cad/files/`
  This requirement can be replaced by SolidWorks *or potentially other CAD softwares. See Section 8. Future versions of FreeCAD may not necessitate this at all.*
- FreeCAD version 0.17
  `https://github.com/FreeCAD/FreeCAD/releases`
- Wings3D
  `http://www.wings3d.com/?page_id=84`
- A CAD model you intend to convert
  - If you are receiving this model from someone else, ask for it in a STEP (`.stp` or `.step`) format.

3 What We Are Actually Doing

In most engineering workflows, we keep track of designs through engineering drawings documenting our parts and assemblies. Recently, these have almost universally become 3D drawings done through Computer Aided Design (CAD) software. In these programs, we model designs as solid objects consisting of various geometric features, and we save these designs as a series of features arranged on top of each other. This is the gist of most CAD formats, but this is inefficient for **42**'s purposes.

Another means of storing these files is as a 3D mesh. In these formats, every object is a finite number of vertices, connected by edges, which are filled in by planes. Objects are formed by arranging these planes to make closed 3D surfaces. Mesh files are what most 3D printers take as inputs (`.stl` format) to generate gcode, and are common in animation applications.

The following process allows us to convert STEP files, a common CAD file type, to Wavefront `.obj` files, a common mesh file type, so we do not have to start our mesh from scratch. We specifically want `.obj` files with an associated `.mtl` file, as **42** uses both.

4 Terminology

In the following discussion, the FreeCAD type body (blue icon) will always be referred to as a "FreeCAD Body". A **42** Body will generally be called simply a "body". The FreeCAD type part will always be referred to as a "FreeCAD Part". The term "part" by itself will generally refer to units of a CAD assembly, and in FreeCAD v0.16 and v0.17, part is closer in meaning to a FreeCAD Body. The term "material" will refer to the visual properties of a complete closed surface, while "color" is used more generally.

5 Guidelines for Initial Models

Your initial model will likely come from someone else. It can be begin in a commercial CAD format if you have the software to open it, or a STEP file format, which almost any CAD program (not **42**) can open. In either case, it may need some edits to work best with **42** depending on your goals. These edits can be done in either version of FreeCAD (though they may be easier in v0.17 or newer versions), or in a commercial CAD software.

5.1 Complexity

In **42**, there is no reason to use high fidelity models. **42** models primarily function as a visualization tool. In fact, the only physical parameter used without a default value is the external projected area for atmospheric drag and solar radiation pressure (SRP).

High fidelity models can only slow the simulation, so if you are starting with a high fidelity model, start by simplifying it. Remove bolts and bolt holes, and any features too small to substantially affect the area calculations. The individual user can decide the threshold for "substantial". If you do not intend to use drag or SRP models, you can simplify further by removing any features not important to your visualization. However, automatic model simplification tools may or may not work, depending on how they handle materials.

As you simplify your model, consider how a `.obj` file is only a collection of vertices, edges, and planes. In particular, round edges and holes will be discretized into a finite number of segments. This means that round features inherently reduce to more vertices than sharp features, so removing them is even more beneficial to your simulation speed. Also be sure to remove any internal features because internal planes cannot be affected by drag or SRP, and cannot be seen by the user. An example of an original and simplified model is shown in fig. 1.

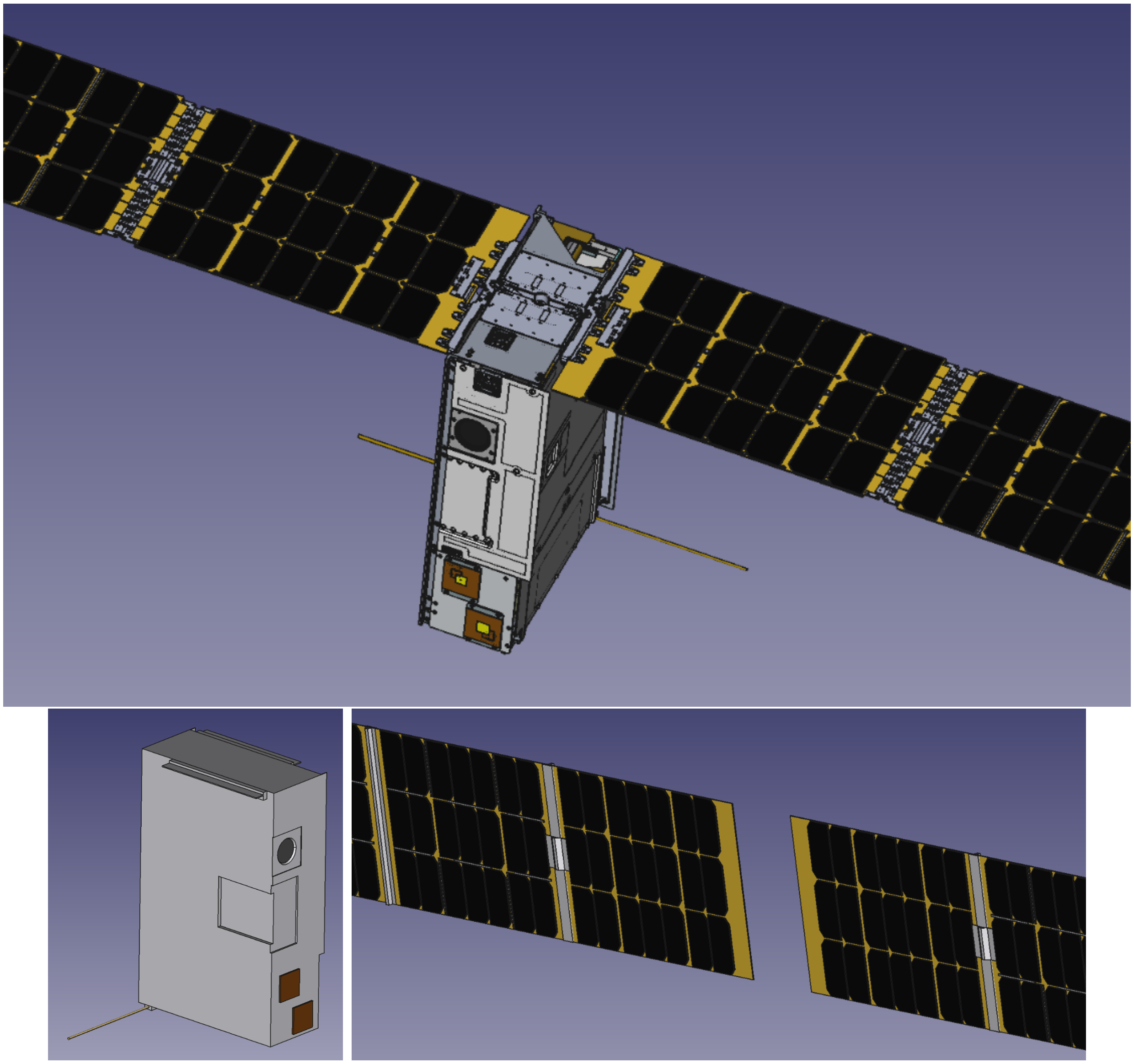


Figure 1: An original high fidelity CAD model and its de-featured counterpart broken into two models. Loading the original model was the most time consuming part of writing this, and will slow down **42** too.

5.2 Materials

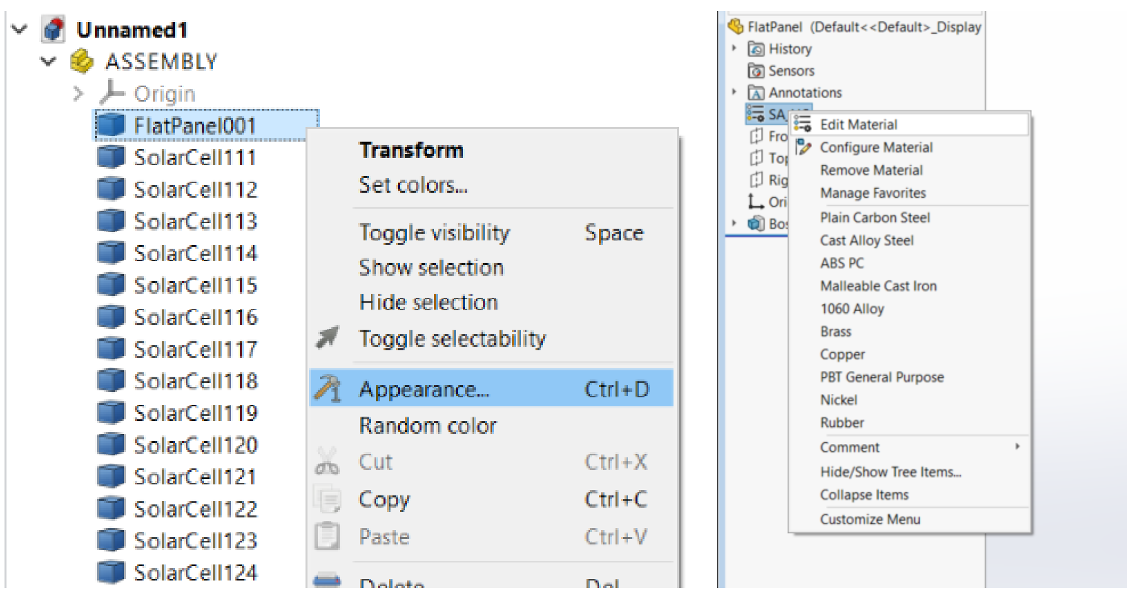
Certain CAD programs allow one to assign colors to each face. However, assignments such as this will not carry through the following conversion process. In the final product, each CAD part must have a single color/material property.

NOTE: It is possible to assign colors to individual faces if you create your model directly in Wings3D, or if you do so during post-processing. However, I try to minimize post-processing in case updates are required. . One can get around this by making a thin part to serve as a face, such as the lens (dark circle) in fig. 1.

Usually, the correct tool to assign materials is called a "Material Editor" or similar. In FreeCAD, the tool is called "Appearance", as shown in fig. 2a—we then change the "Shape Color" in the pop-up box.

NOTE: In FreeCAD, only use the Appearance tool to set materials. The "Materials Editor" in the Arch workbench and the "Set Colors" tool will not work.

NOTE: In FreeCAD, the Appearance dialog box lets you change both the `Material` *and the* `Shape Color`*. Changing the* `Material` *changes both the* `Shape Color` *and the shininess in the display. Two FreeCAD Bodies with the same* `Material` *and different* `Shape Color`*s will export as different materials. Two FreeCAD Bodies with different* `Material`*s and the same* `Shape Color` *will export as the same material, even though they may display differently in FreeCAD.*



(a) Appearance editor in FreeCAD (b) Materials editor in SolidWorks

Figure 2: Materials editor equivalents in FreeCAD and SolidWorks

Assign the appropriate material to each individual part. Then assemble the parts together into what will become your bodies in **42**. In fig. 1, the spacecraft is divided into two bodies, a main spacecraft and a solar array. The main spacecraft body consists of a single aluminum block with cutouts, two square antennas, a nearly flat star tracker lens, and an antenna rod, all different parts with different materials. The aluminum block is solid because its interior is inconsequential.

At this point, do not worry about any material options besides color. Any other material properties will be set later. In fact, you can even set color later if you prefer. In this case, it is helpful to create a placeholder material. For instance, if mylar is not an available material in your CAD program (and you choose not to create it), you can instead make every mylar part some other material with a distinct color, such as bright orange. This material will be easy to identify later, so you can easily replace it with the actual properties of mylar. As you assign materials, remember that **42** does not care about the material strength properties—only visuals matter at this phase.

5.3 How to Divide Assemblies

You are likely starting with an assembly of many parts. Divide this assembly into rigid groups (bodies in **42**), with well defined joints about one another (these joints do not have to be part of your CAD). For instance, in fig. 1, the CubeSat is divided into a main body and solar array body with a one axis gimbal between them. Both bodies contain many parts. In general, bodies for **42** should consist of the largest unit of parts that do not need to move relative to each other.

Save each body in its own file. One way to do this is to delete all parts from the assembly that you do not intend to belong to a particular body. Save this reduced assembly separately and repeat for each body. Often, assemblies have sub-assemblies which make convenient bodies.

As you divide the assembly, keep track of body origins. For each body, you will need to know its center of mass expressed relative to its frame and origin. You will also need the rotation sequences between each body's frame and the distances between each body's origin. **42** will then calculate the net center of mass location, and its variation as bodies move relative to each other.

6 Converting Your Model

The following steps are done once for each **42** body.

6.1 Step 1: Convert to STEP AP214

First, convert the model to a STEP file (type `.stp` or `.step`) version AP214 as in fig. 3. STEP version AP203 may also be an option, but this will not maintain all the properties we want. In FreeCAD, we can do this by selecting every body in the parts tree (use `shift` to select multiple FreeCAD Bodies), and exporting (File $\longrightarrow$ Export) as a "Step with colors" file (same as STEP AP214).

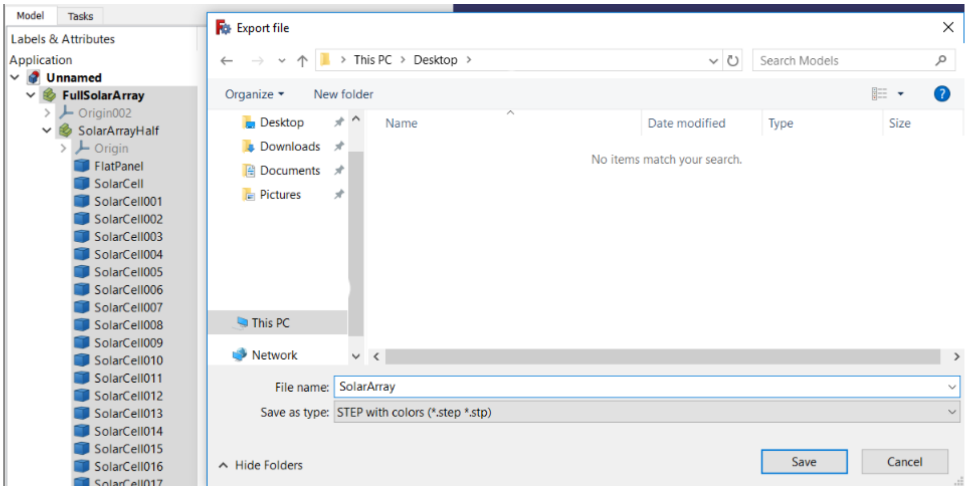
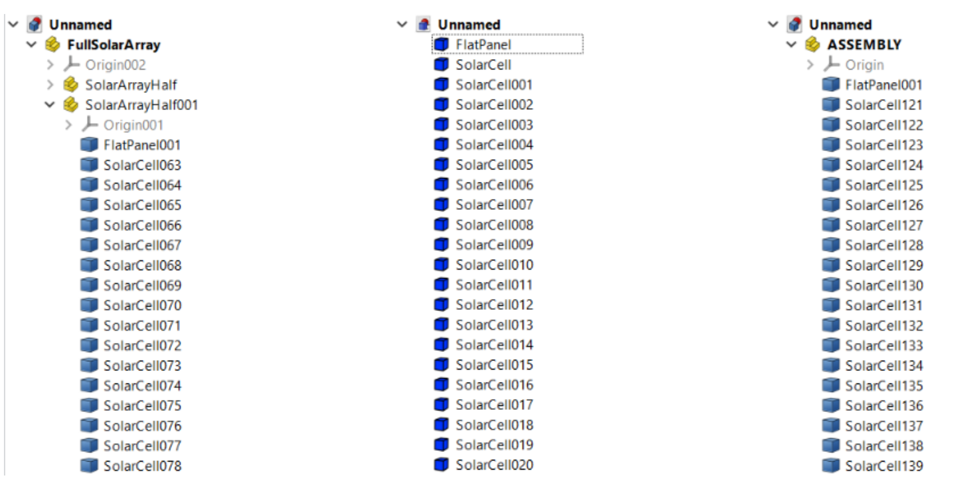


Figure 3: In FreeCAD, save as a STEP with colors file

6.2 Step 2: Flatten Assemblies

FreeCAD v0.17 is still in development and unfortunately is not very good at handling assemblies. For this reason, you need each body to have no sub-assemblies. The conversion process will not fail if you do not do this, but the part locations will not be accurate.

To do this, open FreeCAD v0.16, and open up the STEP file containing everything belonging to a particular body. Note how this version of FreeCAD has no FreeCAD Part object type, so every FreeCAD Body is listed at the same parts tree level in fig. 4b, with no sub-assemblies. Now, re-save this file as a "Step with colors" file. If this new STEP file is opened in FreeCAD v0.17, it will no longer have an assembly hierarchy, as shown in fig. 4.



(a) FreeCAD v0.17 Before (b) FreeCAD v0.16 (c) FreeCAD v0.17 After

Figure 4: The parts tree before and after flattening the assembly hierarchy.

6.3 Step 3: Export to Wavefront OBJ

Re-open FreeCAD v0.17, and open the new STEP file. Under the parts tree, you should see a single FreeCAD Part, indicated by a yellow icon. Expand this part, and there should be several FreeCAD Bodies inside, indicated by blue icons (these are not the same as **42** bodies). If there are instead multiple FreeCAD Parts (multiple yellow icons), then you have done something wrong.

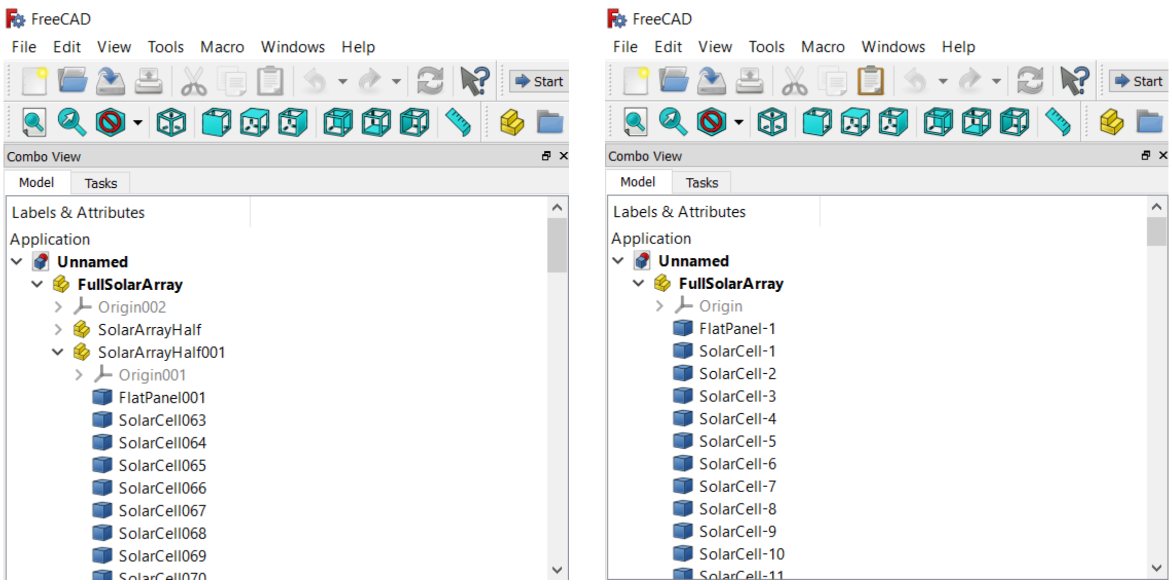


Figure 5: <u>Left:</u> A FreeCAD Model tree with one part containing two other parts. This was exported improperly, and will not work. If this is converted to a `.obj` file, it will only contain half a solar array. In reality, both solar array FreeCAD Parts were exported, but their origins were mixed up, so they exactly overlapped. <u>Right:</u> A FreeCAD Model tree with only one part, containing several FreeCAD Bodies. This is what we want. Select the blue FreeCAD Bodies in the next step.

In the parts tree, select all of the blue FreeCAD Bodies, and export them as a type "Wavefront `.obj`". This should create a `.obj` and `.mtl` file (assuming you are using FreeCAD v0.17 and not v0.16 for this step). Do not save as an "Alias mesh `.obj`" as this will not do what we want. Keep FreeCAD open, or save the file, in case it is helpful for the next step.

6.4 Step 4: Reposition and Rotate

Open Wings3D, and import your `.obj` file. Set your import scale to the appropriate scaling factor (e.g. 0.001 if your original model was made in millimeters), and set your export scale to 1. `.obj` files are unitless, so **42** always assumes one unit is one meter. Now examine your model, and make sure it looks correct. Each grid square in Wings3D is a meter square in 42. You can enter rotation mode by clicking the middle mouse button and exit rotation mode by clicking either other mouse button. You can zoom in and out using the scroll wheel, and pan the camera using the arrow keys. You may need to zoom out to find your model if the model origin is outside the body, or if the camera starts inside the body.

The colors shown in the model at this point represent the materials used in the final result, though their appearance could vary in **42**.

The origin of your model in Wings3D and FreeCAD is also your body origin in **42**. You usually want this aligned with the spacecraft center of mass or some significant point. In many cases, this is not where it is when FreeCAD exports it. If it is, then skip this step.

To correct this, find your reference point in your original model,and record its position relative to some easy-to-find physical feature (like a corner). Find this feature in your Wings3D model and click on it. Above the part, an information pane should appear and tell you the location of that point in the global coordinate system, as shown in fig. 6. Subtract this location from the desired distance from your feature to your desired origin to get a net displacement vector, and record this vector.

> *NOTE: In theory, the global origin is constant in every step of this process, but this document assumes that it may get mixed up. It is often helpful to keep the original model and the Wings3D model open simultaneously, and measure the locations of various features to make sure everything is correct.*

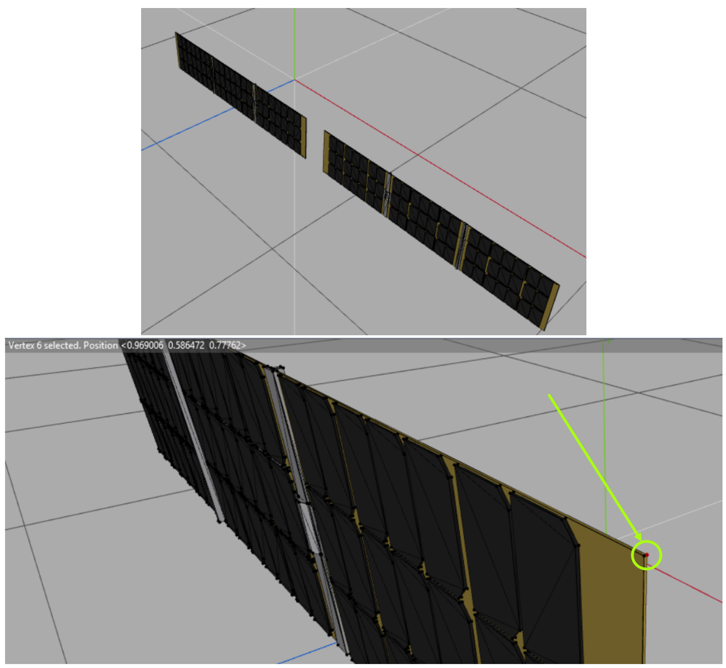


Figure 6: Top: The global origin is the intersection of the red, green, and blue lines, and is not where we want it to be. Bottom: The top right corner of the solar panel is selected, as indicated by the red dot circle, and the position in meters is shown in the top left corner.

In Wings3D, scroll out enough to see the entire assembly. Left click and hold, and drag a box to select the entire model. Once the entire model is selected, click the red shaded box at the top and center of the interface (the rightmost of the four selection options), as in fig. 7. Right-click anywhere in the interface, and then left-click on “Move”. Then left-click on the desired axis. You can now coarsely move the model by dragging the mouse. Press `tab` and enter the appropriate amount in meters to move about that axis. Note the axes may be rotated at this point, so you may need to adjust your vector addition or use a different element of the displacement vector. Repeat with all axes. You may also find it useful to select a point or edge and choose `Absolute Commands` $\longrightarrow$ `Move` to move the model.

> *NOTE: Most Wings3D commands have different behavior depending on which mouse button you use. Mouse over a command and look in the lower left corner of the interface to see what will happen if you click each mouse button.*

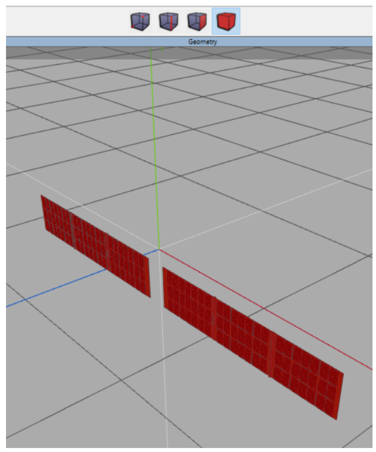


Figure 7: Select everything by holding the mouse and dragging across the model. Click the shaded box to set the selection method to body, instead of vertex, edge, or face.

You now may need to rotate the model, either because the coordinate axes became switched at some point in the export processes, or because you need a particular alignment. With the entire model selected as before, right-click again anywhere in the workspace. Left-click on "Rotate". Then middle-click on the desired rotation axis. You can now coarsely rotate the model using the mouse, or press `tab` to type a specified amount in degrees.

> *NOTE: If necessary, one can also complete these steps by opening the final* `.obj` *file in a text editor, and inserting the comments "*`# Scale up by %f to actual size`*", "*`# Translate by ...`*", and "*`# Rotate via ...`*" at the top, though this is not recommended. These comments are only recognized by* ***42****. Check* `geomkit.c:LoadWingsObjFile` *to see if your version of* ***42*** *supports any or all of these options.*

Wings3D is a great mesh manipulation tool, and there are many more features you may find helpful. However, I have found it difficult to create a model in Wings3D from scratch and prefer using more robust CAD software until I'm ready to start running **42**. At this point, you can also assign materials to individual faces instead of just to complete bodies. Press `space` to de-select everything. Change the selection mode to select faces (third selection option) and click on the faces of interest. Right-click in the workspace, and mouse over "Material". From here, I leave the reader to explore the material assignment options.

When finished, re-export the model as a Wavefront `.obj`. You can now close all other windows, but leave Wings3D open.

`Ns %f`	Shininess [0-1000]
`d %f`	Transparency [0-1] with 1 being fully opaque
`Ka %f %f %f`	RGB Coefficients of ambient color [0-1]
`Kd %f %f %f`	RGB Coefficients of diffuse color [0-1]
`Ks %f %f %f`	RGB Coefficients of specular color [0-1]
`Ke %f %f %f`	RGB Coefficients of emissive color [0-1]
`Refl %f`	Reflectivity [0-1] with 1 representing a mirror
`SpecFrac %f`	Portion of SRP reflected specularly
`DiffFrac %f`	Portion of SRP reflected diffusely

Table 1: List of selected material properties. Note that `Refl`, `SpecFrac`, and `DiffFrac` are specific to **42**, and not part of the `.mtl` file standard.

6.5 Step 5: Adjust Materials

At this point, your model is mostly complete, but your materials file will only contain the most basic information. To improve this, open your `.mtl` file using a text editor. Each material is declared using `newmtl` followed by the material name. In this case, the material name is probably `color_XXXXXX`, which is not very helpful. In Wings3D, at the top center of the interface select the wire frame box with a single face shaded (third selection option). Then mouse over any face, and the material name assigned to that face will show in the upper left corner of the interface. You can use this to identify the materials in the `.mtl` file. Feel free to add comments to the `.mtl` file by starting a line with #.

With the materials identified, you can now add material properties as you see fit, according to the `.mtl` file specification. Generally, this conversion process only keeps the base color of a surface and not any of the specific reflectivity parameters, which you can now apply. Often, I will create custom arbitrary materials in the CAD software, and then apply the exact material properties at the end by editing this file.

For reference, a subset of material properties in order are given in table. 1. A complete list of material properties available can be found by examining the **42** source code `geomkit.h` and `geomkit.c`. Note that the coefficients of specular and diffuse SRP reflection will affect the behavior of the model beyond the visuals. These two coefficients should sum to 1.0 or less. If unset, they default to entirely specular reflection (largest momentum buildup).

For some general guidelines, make sure the transparency is set to 1.0 (or unset, in which case that is the default). If it is set to less than 1.0, then `Kd`, `Ks`, and `Ke` become zero. Test opening the model in **42** using the default material properties, and adjust them as needed. If the model looks off, start by setting `Ka` and `Kd` to the same values and `Ks` to white, and then vary them using the guess-and-check method. Also consider setting `Kd` and/or `Ks` to 0.0 (not their default value). Note that materials will generally not appear in Wings3D the same as in **42**, because of the different light sources.

For reference, ambient color is the basic color of a material when nothing else is present. Diffuse color is an object's color under a light source. Specular color describes an object's tendency to produce glare. Emissive color describes the color an object produces. There are no rules for how to use these options, but they are all available in the visualization engine if one wants to use them.

7 Create Spacecraft Files

Once satisfied with your `.obj` model size/position/rotation, copy your final `.obj` and `.mtl` file to your 42/Model directory. The `.obj` file references the `.mtl` file, and will not open in **42** without it, or if it is placed in another directory. Use these models in your spacecraft file by referencing them by name. Note that the center of mass for each **42** body and **42** joint are referenced to the origins and coordinate systems of your `.obj` files, while the moments of inertia are referenced to the center of mass.

8 Alternate Way to Flatten Assemblies

If the above method is for any reason problematic, parts of it can be replaced. I have not yet found another program which will read a `.step` and export a `.obj` file and a `.mtl` file (usually, only the `.obj` file is possible), so we still rely on FreeCAD for that. However, originally, I flattened assemblies in SolidWorks as follows.

After simplifying and dividing your model, open up the assembly file containing everything belonging to a particular body. Save the file as a Parasolid (type `.x_t`). When saving, click on "Options" and check the box for "Flatten assembly hierarchy", as shown in fig. 8. Now open your Parasolid file. Scroll through the parts tree and you should no longer see any assemblies icons, as shown in fig. 9. You can then export this to a STEP AP214 file and follow steps 3-5 as before. Unfortunately, SolidWork's "Defeature" tool, while extremely helpful, is often not compatible with this process.

> *NOTE: If you have been following this process exactly, but are still getting multiple FreeCAD Parts, restart the process, and close SolidWorks after saving the Parasolid. Reopen SolidWorks, open the Parasolid file, and verify that there are no sub-assemblies left. Save the imported Parasolid file as a SolidWorks type assembly. Then export it to a STEP file. I have not confirmed which CAD programs this works in besides SolidWorks.*

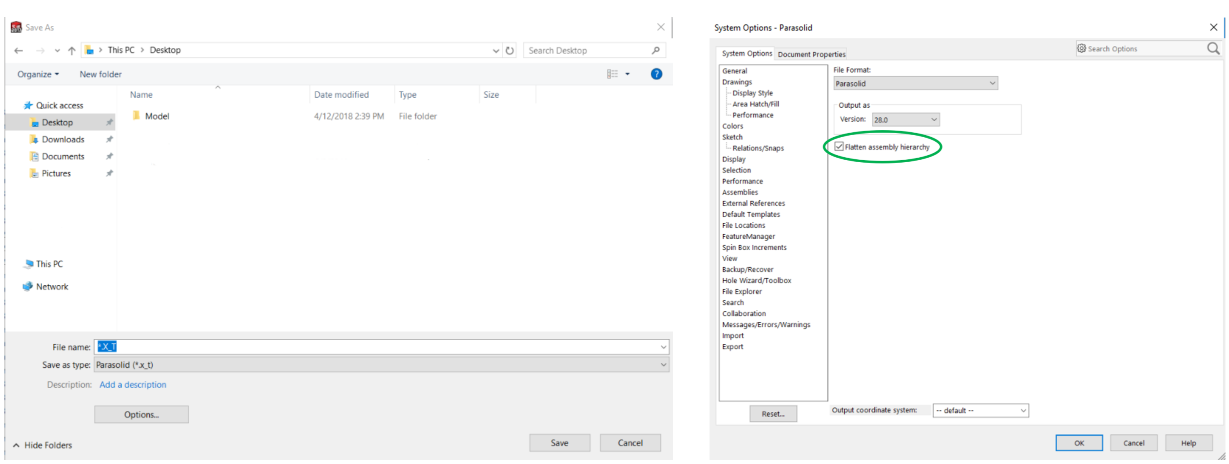


Figure 8: Save as a parasolid with a flat assembly.

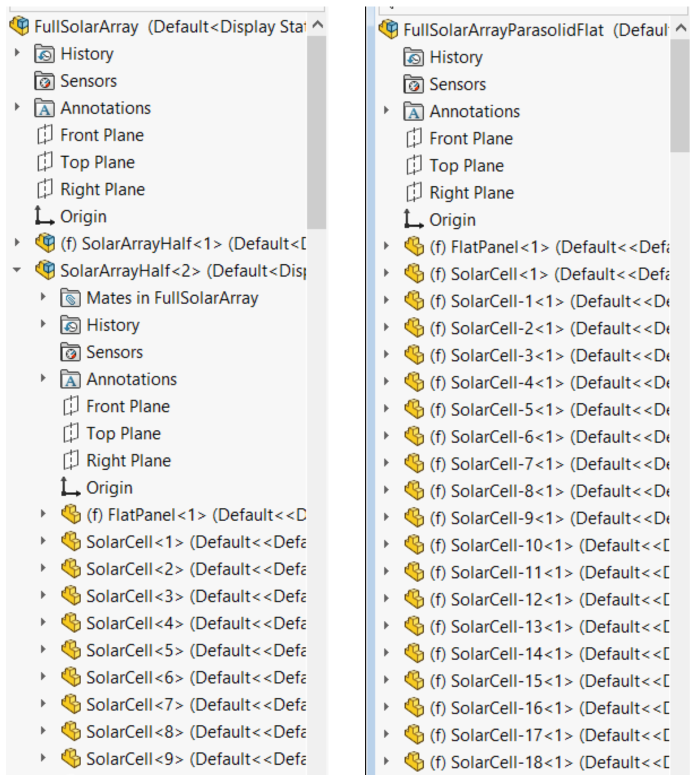


Figure 9: <u>Left:</u> The original assembly with sub-assemblies. <u>Right:</u> A flattened assembly after saving as a parasolid and reloading.