IM00034

Inventor Nesting: What Can It Do For You?

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Learning Objectives

- Discover where Inventor fits in the overall nesting field
- Learn how to establish templates, styles, and workflows for using Inventor with the Nesting Module
- Learn how to set up the nesting template for quick implementation of a new nesting project
- Learn how to create and evaluate comparative nesting studies to find the best nested solution

Description

With the introduction of Inventor Nesting software in Inventor 2019 software, Autodesk made yet another inroad into one of CAD's many niche markets. But what place does Inventor Nesting fill, and how is it best used? Can it replace a dedicated nesting package? What kinds of source geometry can be used with Inventor Nesting? Does it have a place even if a full-featured nesting package is already in use? In addition to a dive into the setup and basic operation of Inventor Nesting software, we'll answer these questions. Come find out what Inventor Nesting can do for you!

Speaker(s)

Walt Jaquith is a Certified Inventor Professional and Certified Autodesk Instructor. A lifelong tinkerer, Walt burned through a first career as a mechanic and fabricator, and then another as a mechanical designer. After getting a preview of the Beta of Inventor R1 software, he never looked back. His current adventure involves teaching and supporting Inventor software, AutoCAD software, and Vault software as an applications expert for IMAGINiT Technologies.

Placing Inventor Nesting in the Nesting Field

Almost any level of operation of a CNC plate or sheet cutting machine will require that part shapes be nested. There are several ways to accomplish this, including importing shapes as blocks into a CAD program and arranging them manually. For anything beyond the most casual use, some sort of nesting software is well worth the expense and effort. There are different ways to get nesting software, and different types of packages available.

Nesting software available to the engineering/fabrication shop generally falls into one of a few broad categories:

Standalone Nesting Packages

Standalone nesting packages aren't necessarily single applications that do only nesting; for our purposes they are separately purchased applications that may truly be standalone or come as part of a manufacturing management suite. At any rate they will not be associated with a specific cutting machine but will be set up to interface in one way or another with any machine the client might have.

Some features often found in standalone nesting packages:

- Job Prioritization and Grouping
- Tracking and scheduling
- Inventory and remnant management
- Integration with MRP and ordering systems
- Direct machine interface (driver library)
- Scalability through options and modules

Figure 1:
CNC cutting machines typically come with a computer controller loaded with nesting software.
Image credit: Rbqlty Machines

Machine-Specific Nesting Packages

Most CNC machines will come with a dedicated software package, and that package usually includes a nesting utility. These applications often run on the computer in the shop which controls the machine itself (Figure 1). Features and licensing may be geared towards that specific machine, which tends to limit the usefulness of the package in terms of a complete fab shop solution. It may fall to the machine operator to perform the nest using shapes supplied by the CAD department. Examples I've seen of this

have been an involved process with several manual steps. Combined with the export/import process needed to get the machine operator the shapes, this makes it prohibitive to run a quick nesting study to gauge material use halfway through a design project.

Free Nesting Packages

My survey did find a few free nesting packages as well. A quick look determined them all to be fairly stripped down, and set to accept shapes in common cad formats, and output nests the same way. While these may be useful for occasional use, it's hard to picture them as an industry-ready solution.

Inventor Nesting

What distinguishes Inventor's nesting module from the others? There are two things to know that will help you place it in the industry in general, and within your own organization. The first is that the only way to get the module is as part of Autodesk's Product Design and Manufacturing Collection. There is currently no standalone license for the nesting module, and no way to add it to a standalone copy of Inventor.

The second is that Inventor's nesting module is not designed to talk directly to CNC machines. Its output is a DXF or MSI file, which is then transferred to the CNC machine. When you consider the applications we looked at earlier, it becomes clearer how this module is positioned in the industry, and it answers some of the questions that come up: Nesting is designed for designers!

Inventor's nesting module is not intended to replace full-featured nesting packages. There is no library of machine drivers, and no option to group parts by work order. On the other hand, if you are running the Manufacturing Collection, the nesting module is available to all your users, and once it is set up it's easy to use, needing very little learning curve. Quick nesting studies are a snap to run any time. This, combined with the fact that Inventor's nesting package is unlikely to be loaded on the CNC machine itself means that it might be a totally different type of user building nests in Inventor.

In terms of feature comparison, the other differences are that the nesting module has some very basic material management capability, and basic reporting functionality, but it is not at the level of the full-featured packages. It really doesn't make sense to have extensive material management in a nesting package which is run on multiple design computers instead of a central machine, connected to ordering and purchasing systems.

With that in mind, let's look at what nesting is, and what it can do.

Preparing Data for use with Nesting

In order to function, Nesting needs four pieces of information for each shape it processes:

- The shape to be cut
- The material
- The sheet thickness
- The quantity of the shape to cut

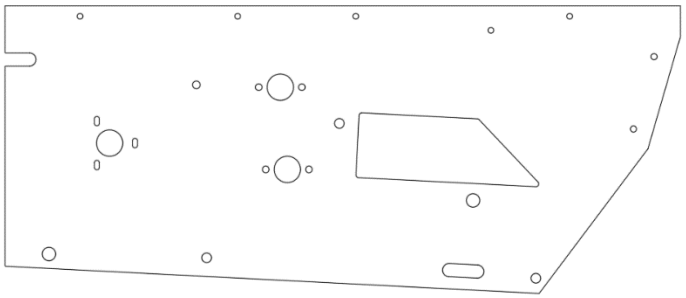

Figure 2:
A DWG or DXF file can supply the shape to be cut, but not the material, sheet thickness or quantity.

Of the types of files which Nesting can process, all supply the shape at the very least, and some types supply more than that. For the types that can't supply all the information, there are provisions in Nesting to add the missing pieces after the import. Here's a look at the types of files Nesting can import, and how easy they are to work with…

Using Inventor Files with Nesting

If an Inventor component file is used as a source for a nesting study, some means will need to be used to determine what shape or shapes will be nested. There are three different ways to use Inventor files with the Nesting Module, and I'm just going to come out and call them the easy way and the two hard ways. The easy way is to create an Inventor sheet metal part file with a flat pattern. The hard ways are to either bring the component in without any treatment, or to author the part files for nesting.

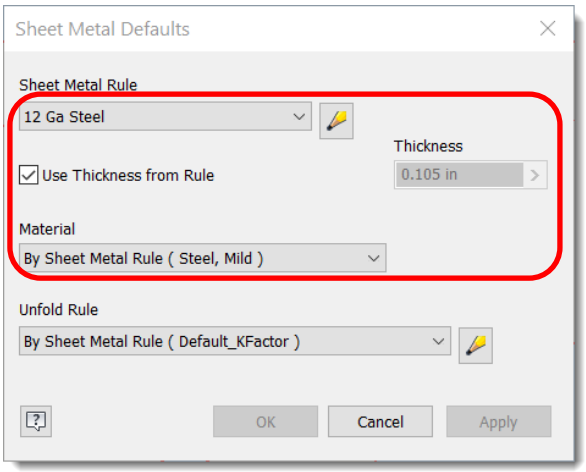

Figure 3:
Inventor's Sheet Metal Styles can provide material and thickness data which can be matched to materials in Nesting's Process Material Library

Inventor Sheet Metal Parts

A prerequisite for any Inventor sheet metal part is that a sheet metal style be applied. Sheet metal styles are normally handled in a sheet metal template, and styles are created there for the different sheet thicknesses, material types and anything else that differentiates one

sheet metal part from another. Sheet metal styles contain all sorts of information pertaining to the bend allowance, corner type, and many other details, but the part that nesting needs is the two main pieces of information in the style: The material, and the sheet thickness. When a sheet metal part is imported into Nesting, the module will look for a match for those two properties in its own material library. If it finds matches, all is well. If not, it offers to create a style based on the import.

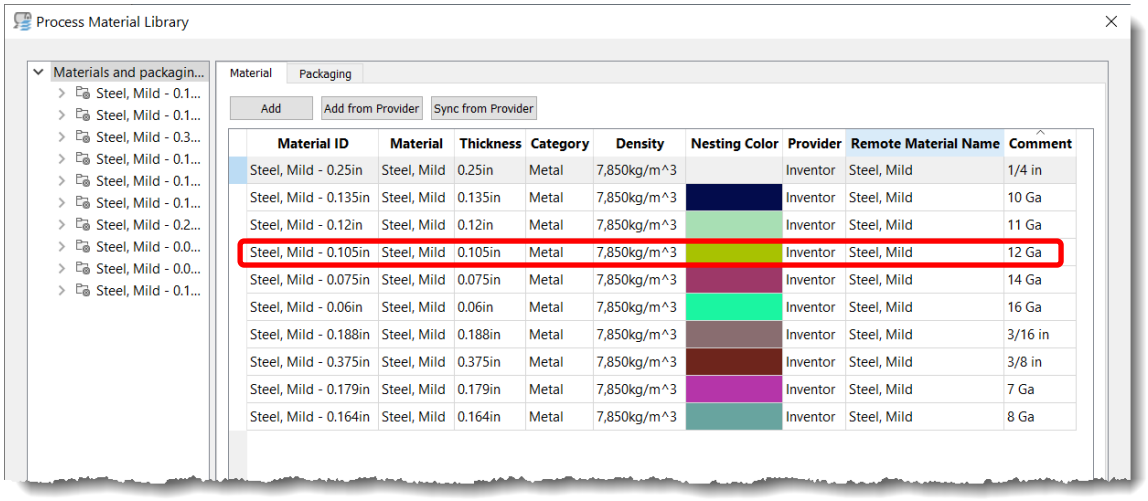

Figure 4:
Nesting will try to find a material and thickness match for an imported shape in its Process Material Library. If it can't, it will offer to create a new material for that shape.

Besides the material and sheet thickness, two more pieces of information are needed; the shape of the part to be cut, and the quantity. The flat pattern provides the shape, and if the part is in an assembly, then the assembly supplies the part quantity. This means that an Inventor sheet metal part in an assembly is the one source for nested shapes which can supply all the information Nesting needs to do its job. This is the best and easiest path for doing quick nesting studies in Inventor.

Managing Non-Sheet Metal Parts

Excluding shapes from nesting

Before exploring how Nesting handles non-sheet metal components, it's worth considering how to exclude parts from the nest if needed. We have already mentioned that regular (non-sheet metal) parts from Inventor can be brought into the nesting environment. When you think about what a nesting package needs to do its job, and what it's designed to do, it should be obvious that just bringing any

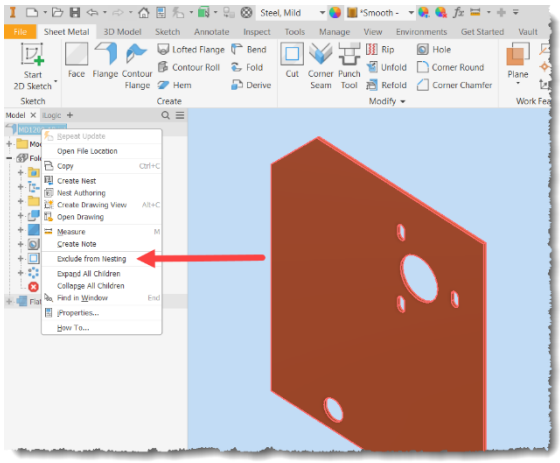

Figure 5:
Excluding a part from Nesting

random part into Nesting may return nonsensical results. That is why Nesting allows us to exclude non sheet metal parts using the Providers dialog box. This is the first option for telling Nesting to ignore certain file types, but if the all-or-nothing approach doesn't fit your workflow, there are other ways as well. Parts can be individually excluded from nesting by checking that option for each part in the right-click menu in the browser (Figure 5). Entire sub-assemblies can be excluded this way by excluding the sub-assembly itself. Of course, this requires more direct management by the user, but it does offer workflow options that would be hard to get any other way.

The final way to exclude a component in Inventor from nesting is by tagging it's BOM structure as "Purchased". Nesting will ignore all components which are tagged this way.

Nesting Inventor Non-Sheet Metal Parts

If the option is checked to allow regular Inventor parts, and a regular, unprepared Inventor part is imported, some criteria will have to be used to determine the shape and thickness to be nested. Nesting chooses based on this:

- First it looks for a visible, unconsumed sketch in the last node in the browser tree
- Then it goes back to the first extrusion in the part and uses that

For the thickness, Nesting will use the length of the first extrusion if it can, and if not, it uses a default .120 thickness, which can be changed later if needed.

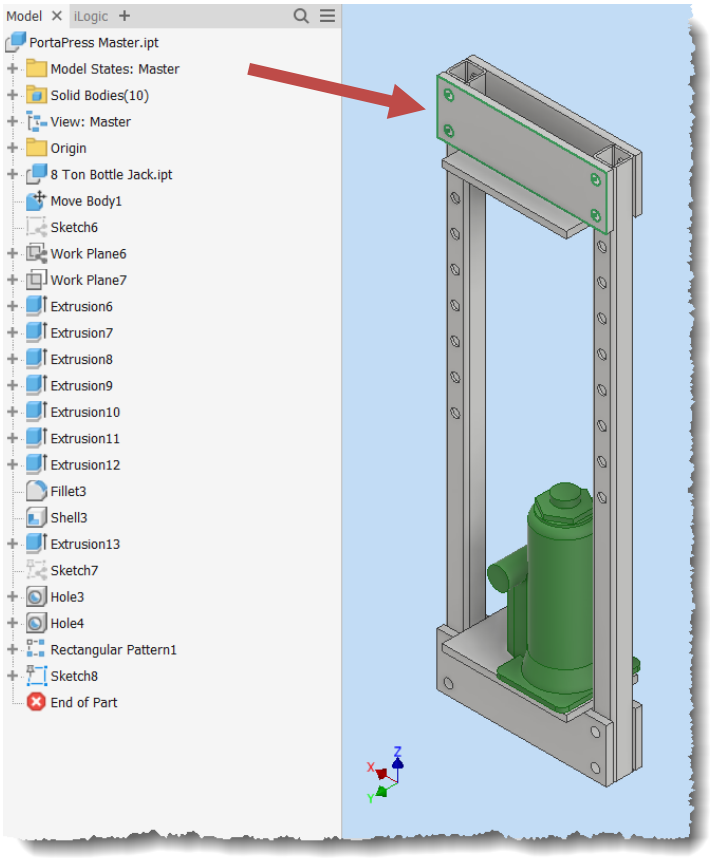

Figure 6:
If this non-sheet metal, multi-bodied part is nested, Nesting will use the visible Sketch 8 for the shape. If no sketch is visible, it will attempt to find a shape in the first extrusion; in this case Extrusion 6.

From all this it should be evident that a non-sheet metal Inventor part would require specific set up and attention to the browser tree to return a good result in Nesting. While this is a viable workflow for certain parts, it does not have the ease or natural flow of a sheet metal part with a flat pattern.

Other Types of Files

Files brought in from other CAD programs can be saved as Inventor files, and then nested. There will be no Inventor features in the browser tree, so instead Nesting will look for the largest planer face it can find and use that as the shape to nest. If there is a discernable thickness perpendicular to that shape, it will use that for the thickness. Missing information such as material and part quantity can be added once the file is imported into Nesting.

Authoring Parts for Nesting

Nesting information can be embedded in a part file by right-clicking on the part icon in the browser and selecting "Nest Authoring". Depending on the part and what types of features it has, Nesting will do it's best to determine a planer shape for the nest, and a thickness based on the part's extrusion length perpendicular to that face. In the example in Figure 7, it picked the flange profile as the largest planer face it could find. The overall length of the part (over 4 inches) was picked for the Nesting Thickness. I unchecked that box and added the 0.25" thickness myself.

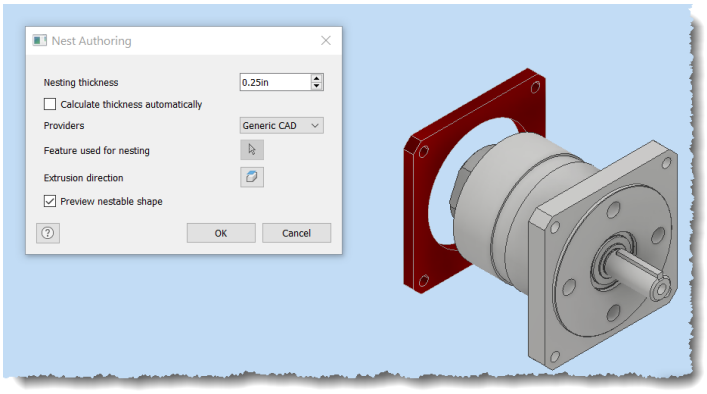

Figure 7:
Authoring a part for Nesting

Setting Up the Nesting Environment

Nesting creates a separate file with a .inest extension. A template file is used to maintain environmental settings. As with other Inventor file types, a good template goes a long way towards insuring a good experience with nesting.

Common Template Settings

The Providers Dialog

Providers are set from the flyout in the Manage panel (Figure 8). The settings are straightforward, and many of them are simple defaults which cannot be changed. The only thing to worry about here is the IPT file settings. This determines whether Nesting will bring in various file types, and the order it will work on them. Note that while it does list "Generic CAD" as an option, those files will need to be saved as Inventor IPT files to be imported into Nesting. With the providers set to best reflect your planned workflow, we're ready to set up the materials and packaging.

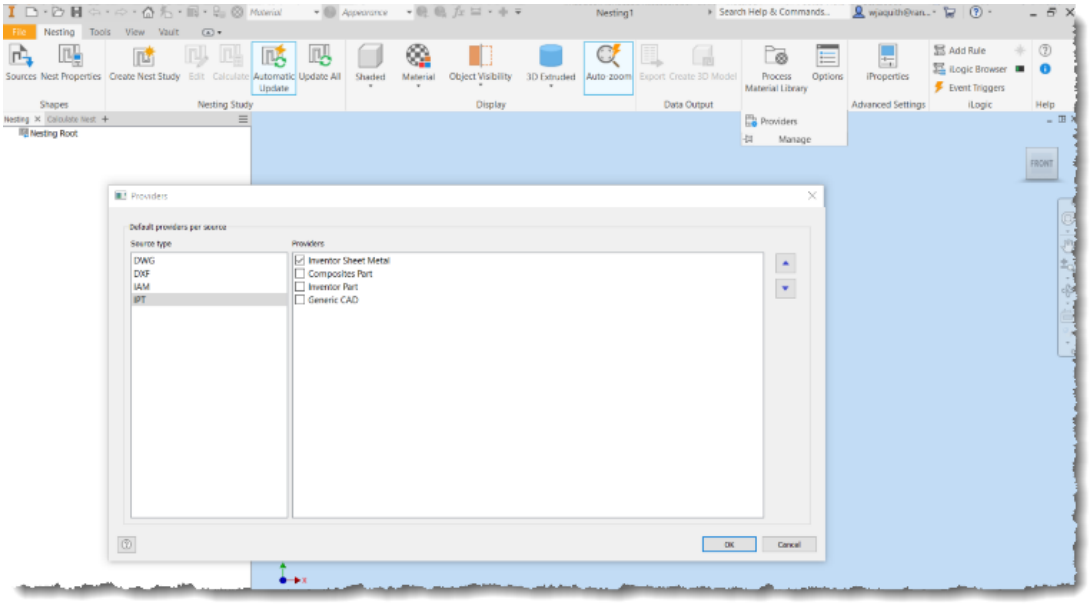

Figure 8:
The Providers Dialog Box

The Process Material Library

In the Process Material Library (See Figure 4) we set up combinations of material, thickness, and sheet sizes (Packaging). When Inventor sheet metal components are imported, Nesting will look for a match between the materials here and the ones in the sheet metal styles. If it finds a match it will use those materials. If not, Nesting will offer to create a material based on what it can glean from the imported part.

Defaults can be set for each material for what rotation angles are allowed, how much material should be left between each part, and the angle of the material's grain. Once materials are created, they can be assigned packaging, which involves information about the sheet size—which can be rectangular, or in a roll—and a set cut distance from each edge if desired.

Nesting Options

The Options Dialog has some settings for appearance, color, and such, but the real story here are the configuration files. The extract and export Nesting functions are both governed by configuration files, and the default configuration file is read-only. Any number of

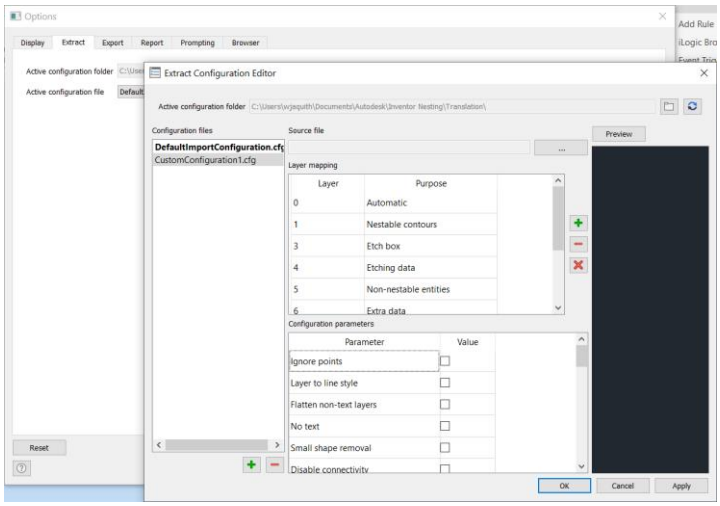

Figure 9:
A custom import configuration file

custom configuration files can be created, where layers can be mapped and other options set. For general use with Inventor sheet metal parts the default files will likely be fine. However, if you are importing other files such as DXF and DWG files, and those files are set up with the shape information on a certain layer, a configuration file can be created to match that setup so that Nesting isn't trying to pull non-nestable data in with those files.

For exporting the nesting study to a DXF file, the configuration file determines what layers the information goes on, same as with the import. If a custom configuration is needed, this is where it is created.

Creating a Nesting Study

Once the Nesting environment is set up, and the files to be nested are prepped, it's time to create a nesting file, import the shapes and run the nesting study. A new nesting file can be created from the template as with other Inventor file types, or by right clicking on a component in the browser and selecting "Create Nest"

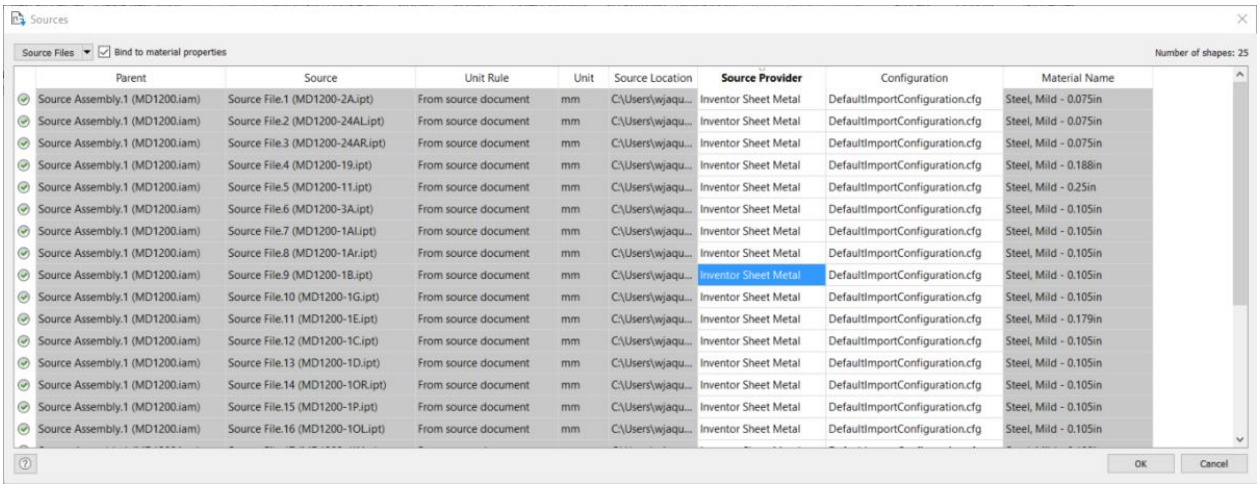

Figure 10:
The Sources Dialog Box

Importing Files into Nesting

The first step in creating a nesting study is to import the files which contain the shapes you want to nest. Any number of sources can be combined in a single nesting file. The obvious choice for a nest is Inventor assemblies containing sheet metal parts with flat patterns, but AutoCAD DXF and DWG files as well as regular Inventor part files and part files converted from other file formats are candidates as well.

Selecting Sources from the Shapes Panel in the Ribbon, and then the Source Files tool allows the user to select the files for Nesting (Figure 10). Depending on the Providers setting, and the types of files imported, there may be additional information to provide in the Sources dialog. Anything not specifically provided by the source will be editable. If

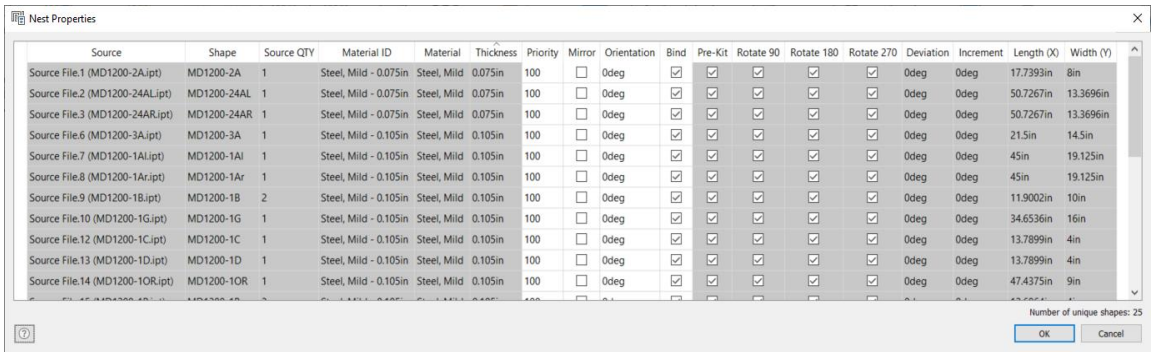

Figure 11:
The Nest Properties dialog box

matching materials can't be found in the Process Material Library, nesting will offer to create the missing materials and sheet thicknesses.

Once the files are brought in, Nesting displays them in the browser, and lines them up in the work window, color coded by their different materials.

Prepping the Shapes in Nesting

Selecting the Nest Properties tool (Figure 11) will bring up the Nest Properties dialog box. Here Individual shapes can be given specific instructions about how they will be nested. By default, these settings are bound to the settings which were set when the material was created, but they can be unbound and managed individually here. If there are any parts that need controlling as far as how they are placed with the grain of the material, or if the shape needs to be mirrored, here is where it can happen.

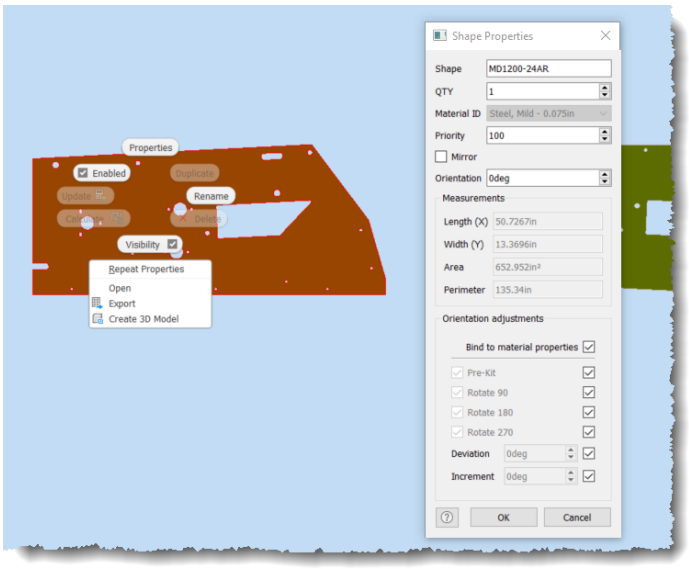

Figure 12
The Shape Properties dialog box is used to access properties for individual shapes

Another way to get to the shape properties is by right

clicking either the part in the graphic window or the browser and selecting Properties. This brings up the Shape Properties dialog box where changes for that shape (Figure 12). This dialog box also allows me to override the source quantity and specify how many of that shape should be cut.

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Creating a Nesting Study

With all the materials assigned and the properties sorted, a nesting study can be run. There are several tabs of parameters that can be set here. Some are standard, and most are self-explanatory. Some important points:

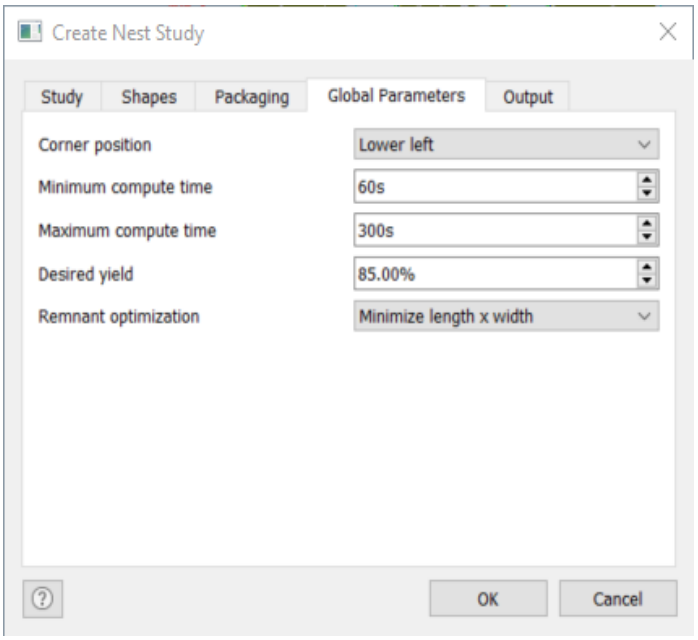
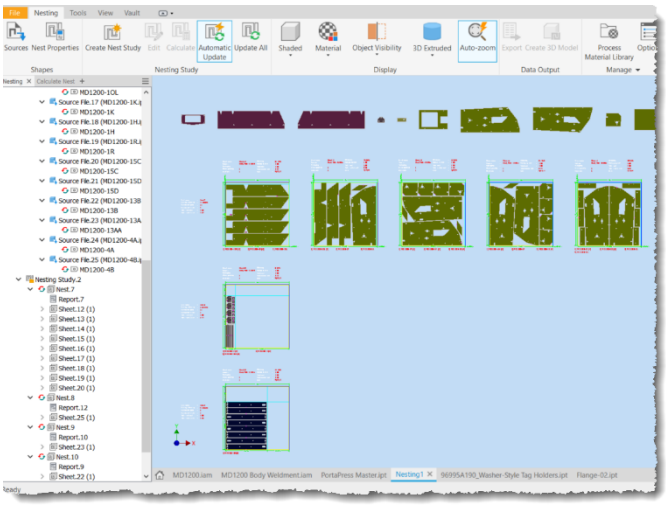


Figure 14:
The Create Nest Study Dialog Box

- On the Study tab the study can be named, and the job quantity set. If there is more than one top level assembly in the nest, quantities can be set for each one.
- On the Packaging tab, different packages can be added, and a preference set as to how the packaging is prioritized.
- On the Global Parameters tab, the desired yield and min/max runtimes can be set, as well as a preference for how the remnant is optimized.



The nesting study is created and placed in the drawing view below the shape previews (Figure 15). If a second study is run it will be placed under the first. Any number of studies may be run in a single nesting file. This is useful for changing the nest parameters to see how the nest is affected.

Figure 15:
The Nesting Study

Exporting The Nesting Study

Inventor Nesting does not communicate directly with CNC machines. The sheets are exported to DXF or MSI files by right-clicking on them in the browser and selecting Export. An opportunity will be given to select which configuration file should be used for the export. By default, the shape and text info are separated onto different layers.

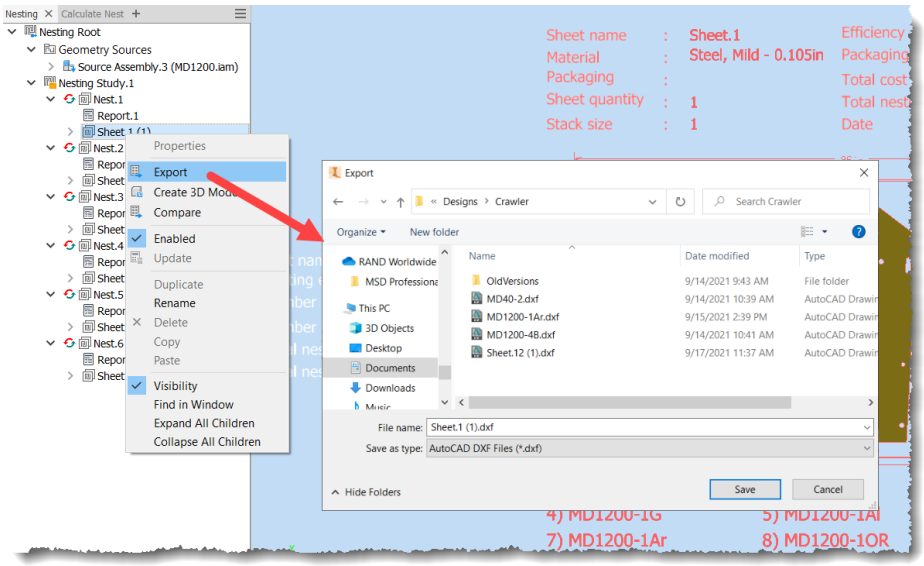


Figure 16:
Nested Sheets are exported individually to DXF or MSI files

Comparing Nesting Studies

If a second study is run in the same nesting file, Nesting will place it under the original study. Andy changes made in the nesting options dialog will be reflected in the new study. In this way, nests can be compared to see the impact of the changes.

Wrap Up

Inventor's nesting package is designed to bring quick, easy nesting studies to the engineer's desktop. With minimal setup and good integration with normal Inventor sheet metal workflows, Nesting fills an important niche in the nesting world, offering functionality and convenience that's hard to find in other solutions. While it does not attempt to compete with industry class nesting packages, it works well along side them, freeing up the heavy lifters from the need to run intermediate studies for estimates and scheduling. For anyone using Inventor's sheet metal tools, nesting will be a valuable addition to their toolbox.