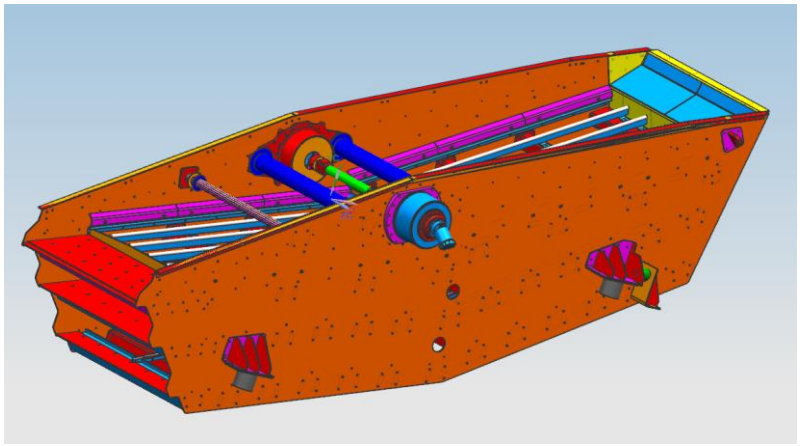
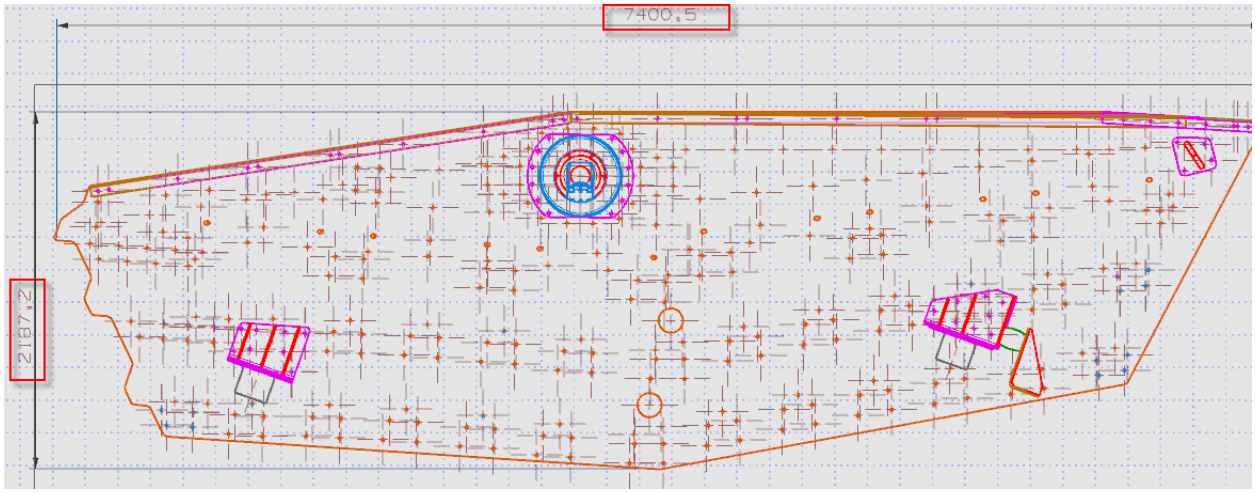
Hole definition in a flat pattern DXF file

Holes are In general a difficult topic. You need them in a DXF file to define how the geometry will look like after flame cut, plasmacut, lasercut or other manufacturing procedures.

NX has only one option. put everything that exists in the model and include it to flat pattern. (up till NX11).

This is not optimal in many ways. Lets take an example of a screen side plate. It has with a careful estimate more than hundred holes. Most of these holes are big enough to be cut with laser. Some are big enough to be cut also with fine beam plasma. Some might be plasma cutted. Only a few can be flame cutted. Some have to be drilled and cannot be cut. Picture of a screen and it's side plate below.





The size alone sets some restrictions to manufacturing methods. Not to mention how many different methods might be used. Cutting, drilling, machining…. How do you know which you can cut with laser??? Nightmare for a nesting operator. Plus all holes are needed in the end anyway.

Every company with efficient global procurement processes needs to rely on standardized output towards their supply chain. Modern partners and less capable developing market suppliers must be able to use the same material from the customer. This means that while there could be 30 suppliers worldwide supplying the same items, the documentation cannot rely on tools or processes from one supplier. This means general documentation with all needed info in it.

How does this become a problem in NX?

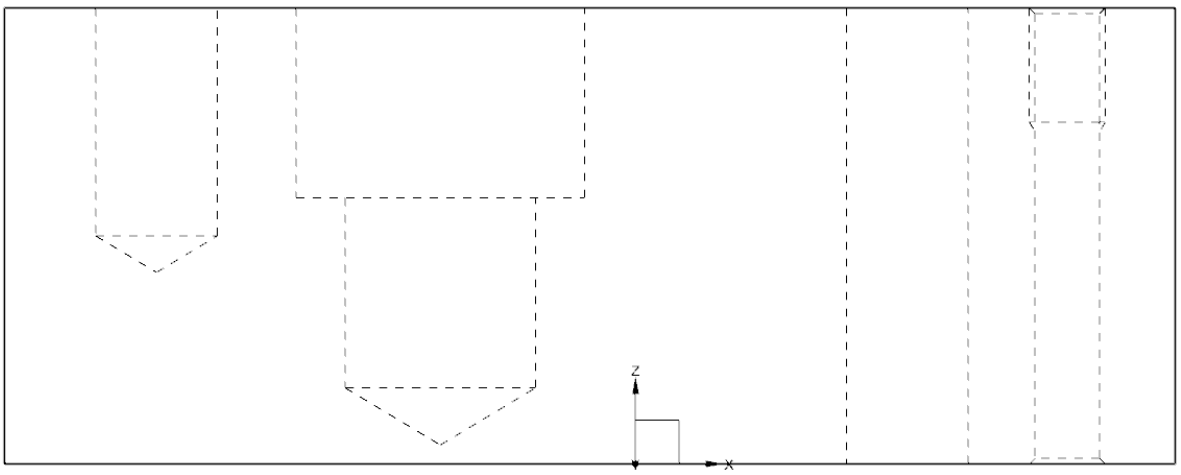
Well, first problem in NX is that flat pattern does not recognize holes as separate features at all. this means all holes are defined as internal features or internal cutouts..

When one exports a flat pattern, all holes are included. Regardless of the type of hole or the size. And regardless if they can be cut or not.

What is a hole type in this context? Interesting differences could be defined as follows:

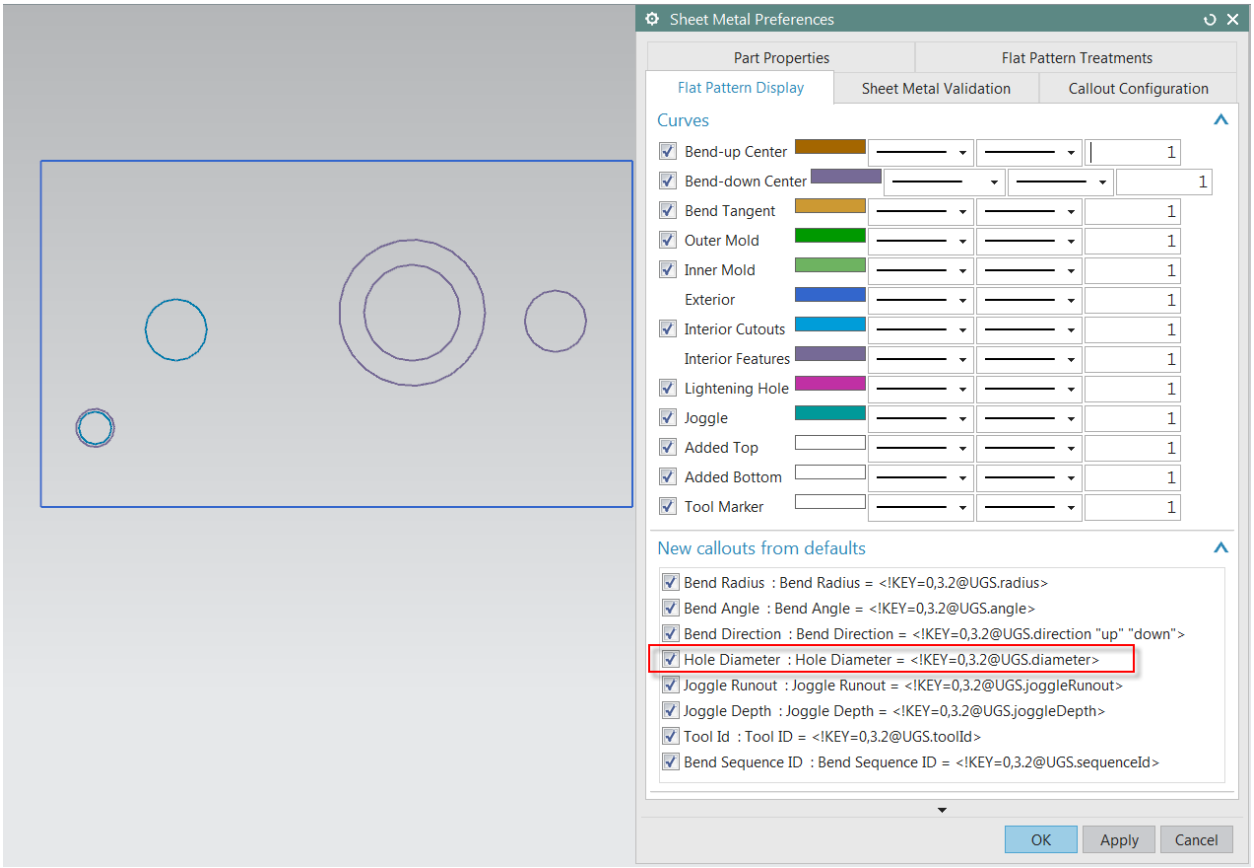
1. regular holes that go through the plate. (can be cut with laser, plasma, or flame cut. Can also be drilled, machined ect) this is interior cutout.
2. Tapped holes. Cannot be cut without adjusting the hole geometry in DXF file to get the appropriate size to accommodate thread.
3. Precision or tolerance holes that go through the plate. Can only be drilled or machined
4. non through holes. Can be a regular hole, tapped hole or a machined hole. (Interior feature(

Below an example of few different holes. When exported to a flat pattern DXF they all are internal features or internal cutouts. There is no way to separate different types with their own export layers per hole type.



Some hole geometry is internal features and some interior cutout.

Everything will nevertheless end up in flat pattern. Without the possibility to separate to a different layer. Interior cutouts seems to be a layer to have all through geometry without any machining features.



Lets get back to the business problem. To get things cheaper and faster. Need new suppliers and need them cheaper. This leads to optimizing (minimizing the work) that the suppliers need to do to quote and manufacture. Different suppliers have different technologies and methods. Still they need to be able to do what they are needed with the same material provided by our procurement. Of course none of the suppliers like to put in extra effort to maximize their earnings.

One statistic was that most suppliers use 20% of their working time making quotes. When nobody has the real time nor money for that, those quotes become guesses. Either they are good guesses or not. Anyway, cost of that labor usually needs to be transferred to the customer side sooner or later.

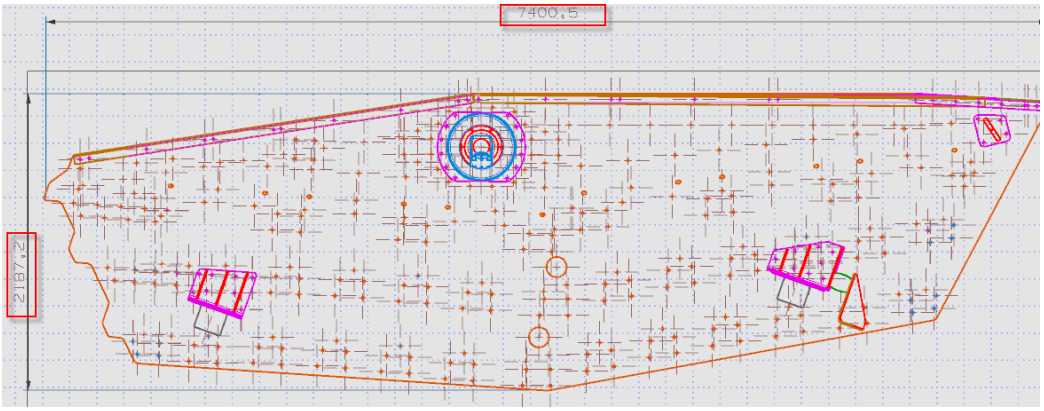
What does this mean? It means that with sheetmetal parts the DXF is a standard file format that even the most old school supplier and developing country company can usually open. So the DXF (flat pattern export) should have everything needed for the first steps in manufacturing the sheetmetal part.

So by stating that all geometrical aspects should be in the flat pattern we present a problem. How to get everything in without complicating the file too much. Answer is to have separate layers for different things.

Take holes for example. Suppliers can easily hide holes that need to be machined if they exist in the file with a distinctive layer.

So would be the case with tapped holes. Production planner can easily turn off everything else and modify the diameter for only those holes that have threads. Or choose that they are too small to be cut anyway with that companies machines.

Currently If all holes are with the same layer, one can imagine how easy this part would be to plan for nesting…



Another thing to consider. Most cutting methods have a limit to hole size that can be cut with a certain process. These fabricators seem to often replace the hole geometry with a center mark in their cutting. This center mark can be etched with laser to represent the center point in a later drilling and tapping process. So to have a simple choice to include a center mark to all small holes deemed non cuttable (simple formula that could be defined in sheetmetal preferences like put center marks to all holes that have a dia. Smaller than plate thickness for example.

If it would be possible to have different layer groups for holes, it would make sense to include the center mark in the following layer to the hole. Example, tapped holes=20, tapped hole CL=21.