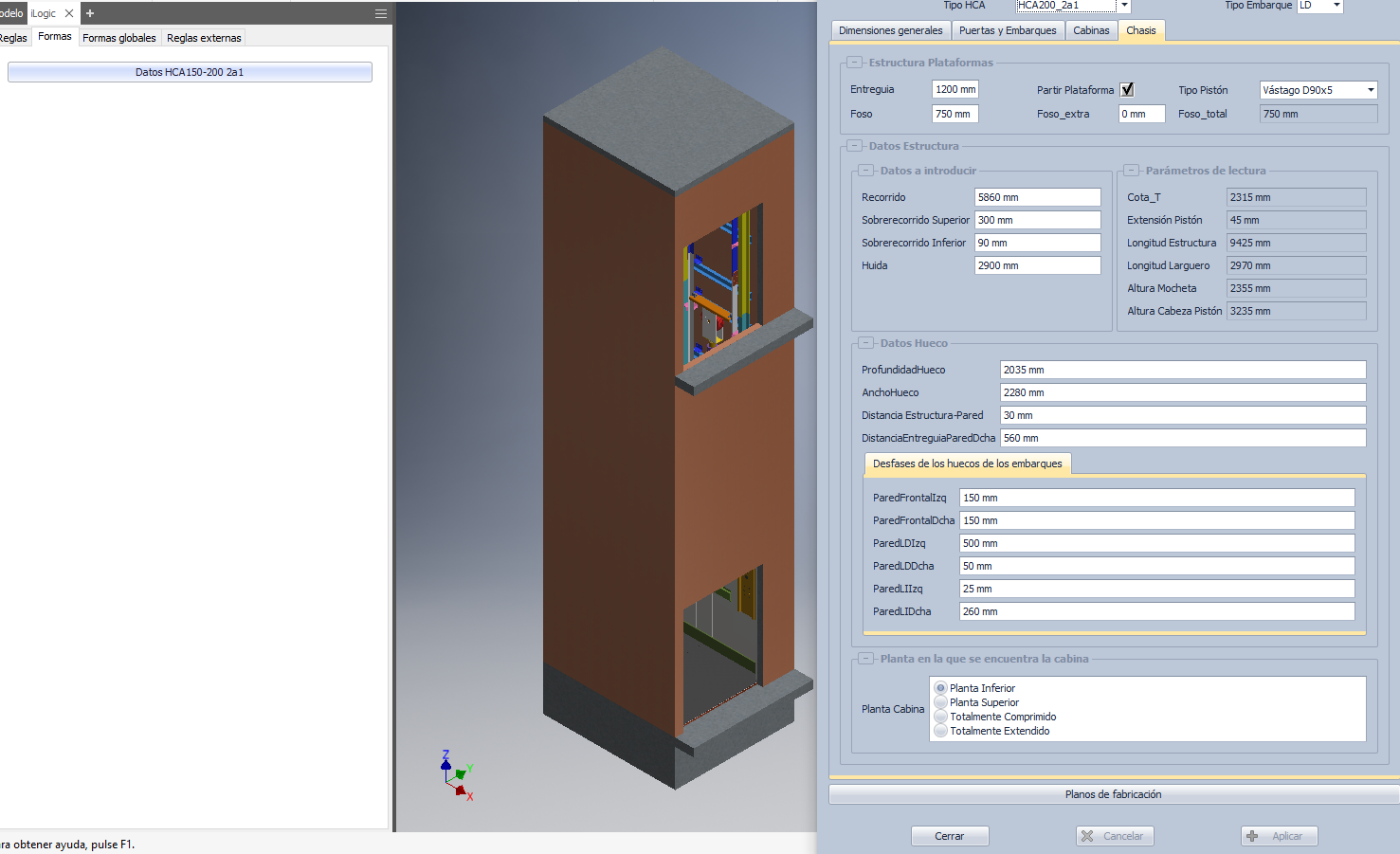
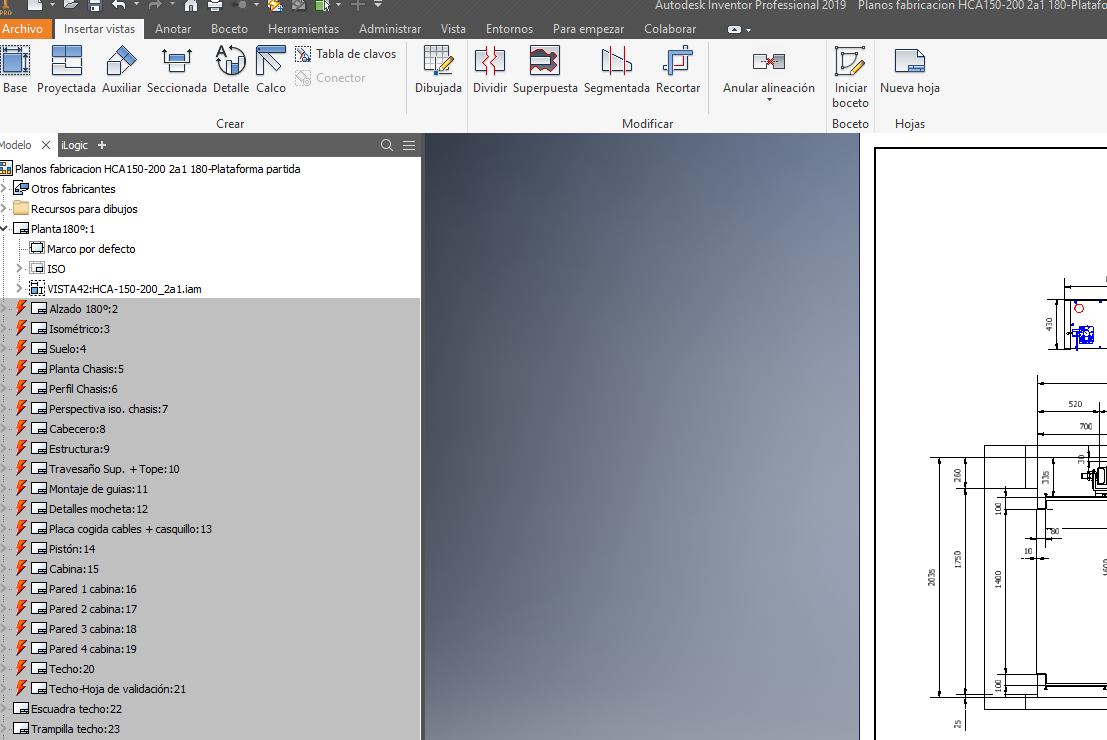
First of all, I put the dimmensions and options that I want, ruled by ilogic, in a form like this:



Then, I wait a few seconds, and the assembly automatically change to the one I want.

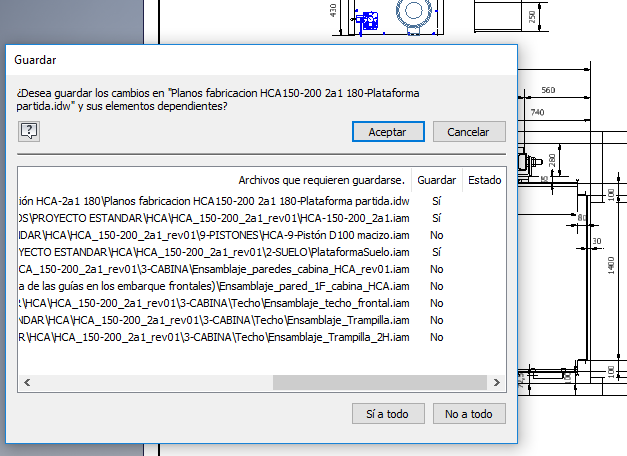
Then, I open a drawing in which I have differents sheets and I update all of them:



I update the drawings:



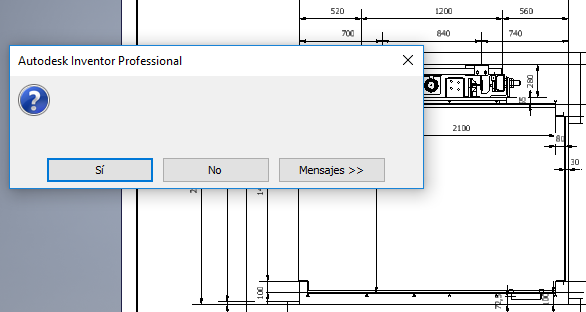
Then, I save the Project in this way:



Note: If I click on “No” to save the “.iam” parts, there is no problem. But I prefer no to do it every time, and just not change the “.iam” model when I save the “.idw”.

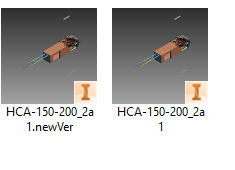
Up to this point, I get everything in a correct manner. I get my drawings and everything runs ok. The problem starts when I save the drawings and I reopen the model.

After that, If I close the .idw, Inventor show me this message:

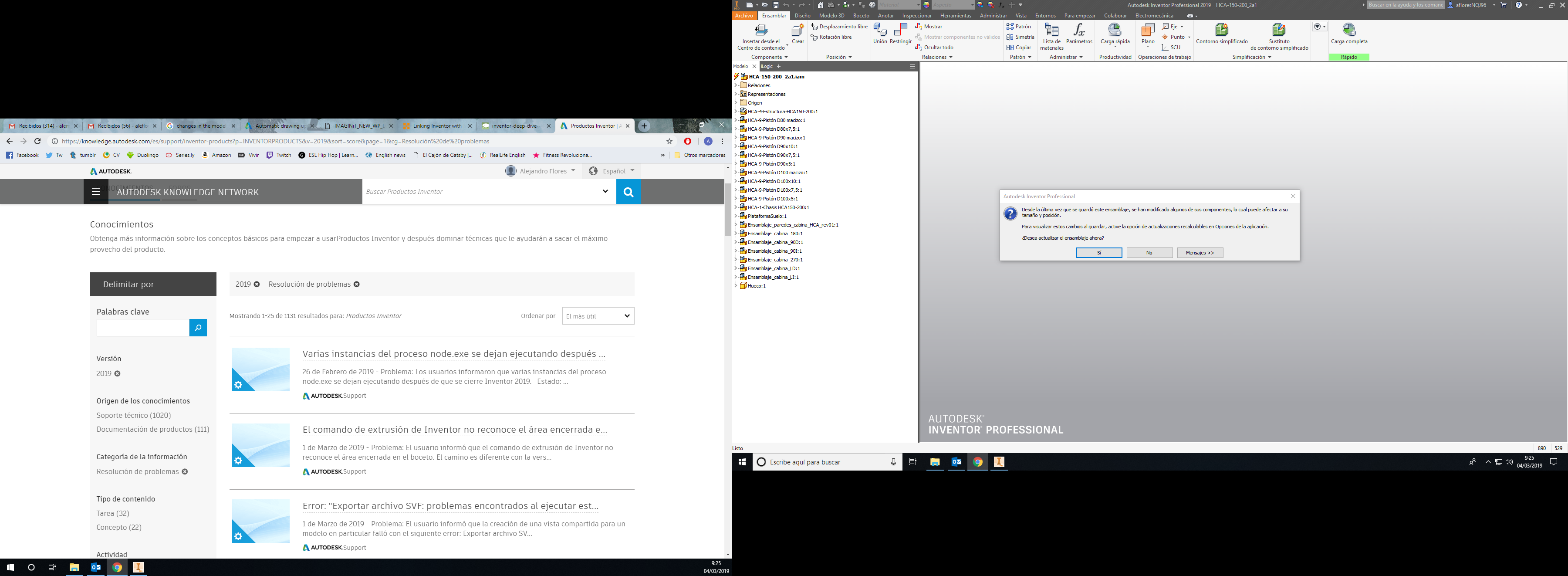


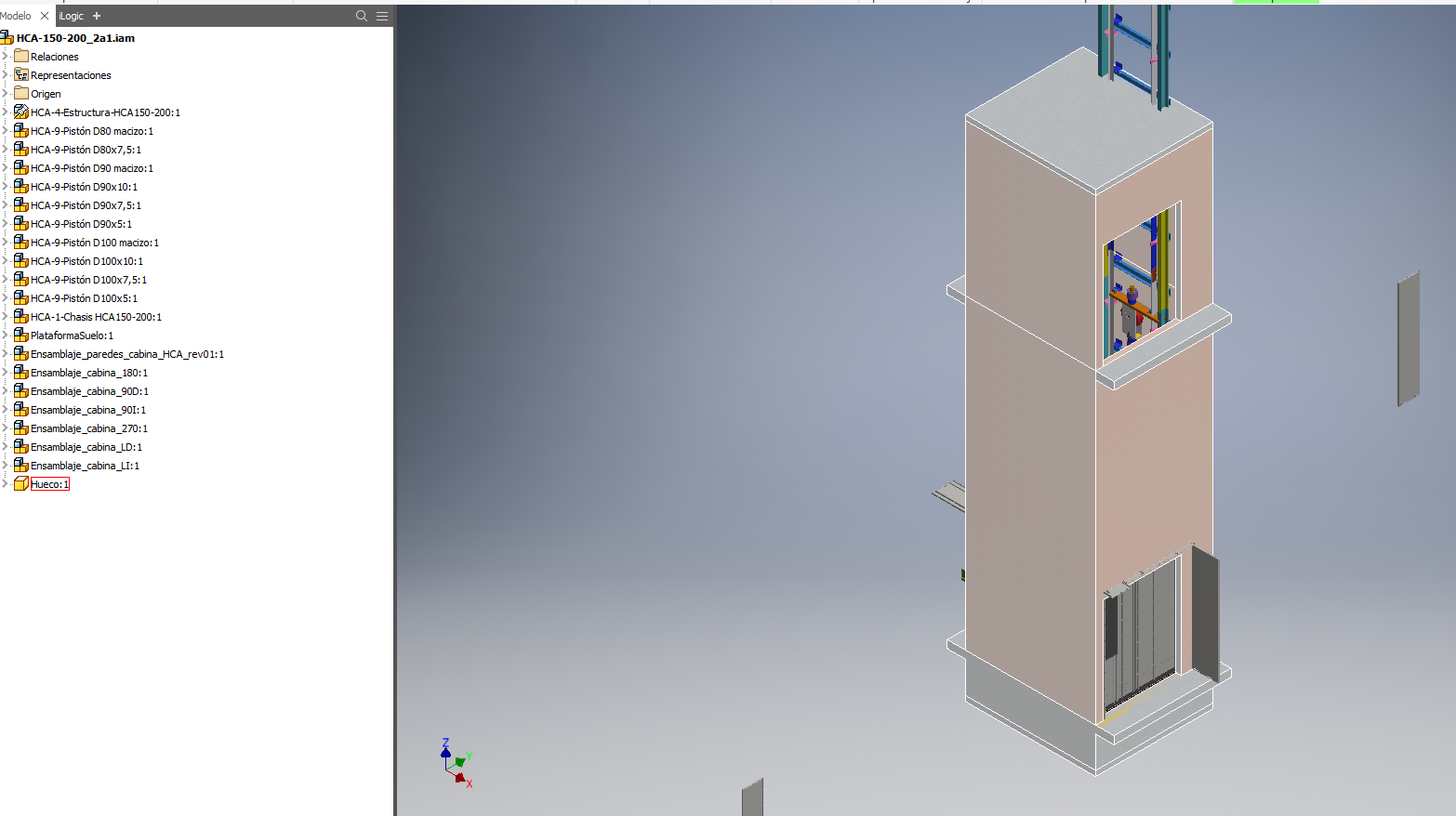
And here I dónt know what to do, so I just click “Yes” (sí).

I found in my folder that now I have two .iam of the model, cause I have to update it…:

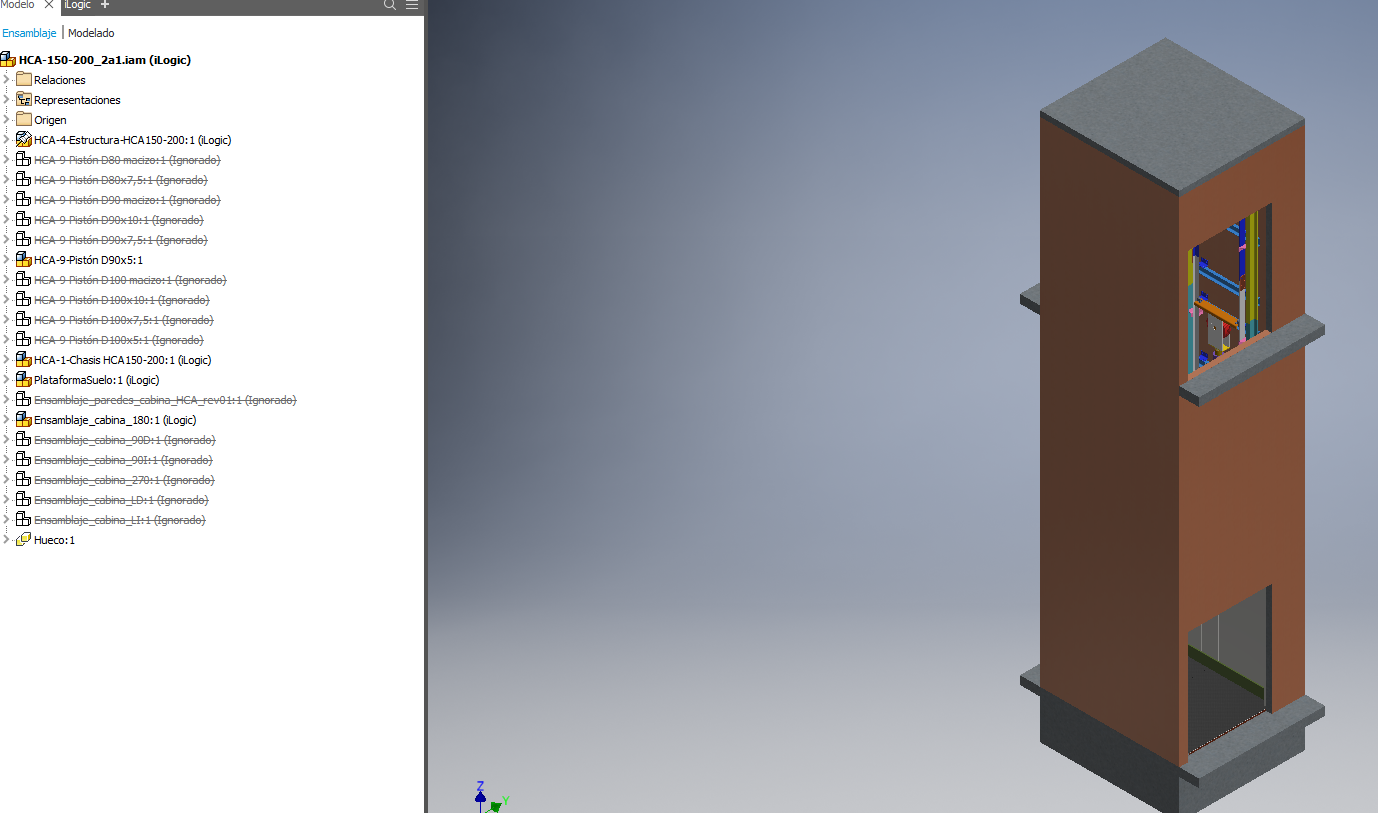


If I reopen now the model, I have this question about updating the model, and I found that all the differents sub-assemblies that are commanded by an iLogic rule are now in the “Master” Level Of Detail instead of the “iLogic LOD” that they are supposed to be:

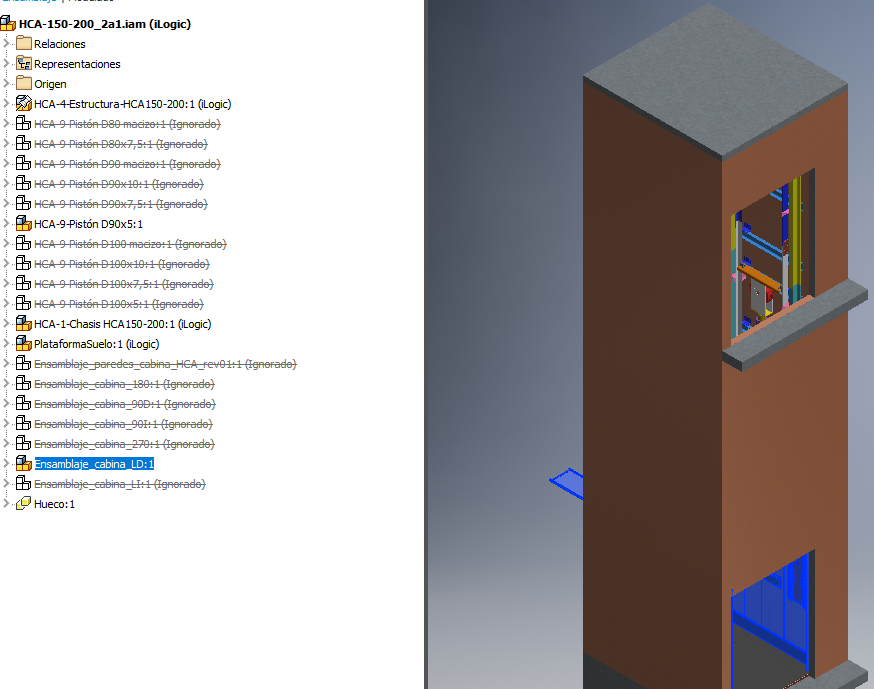


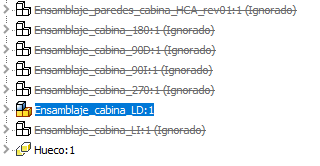


I mannually change the level of detail of the master iam to the “ilogic” level and I manually run too a rule that put me the correct assembly, parts and dimmensions:

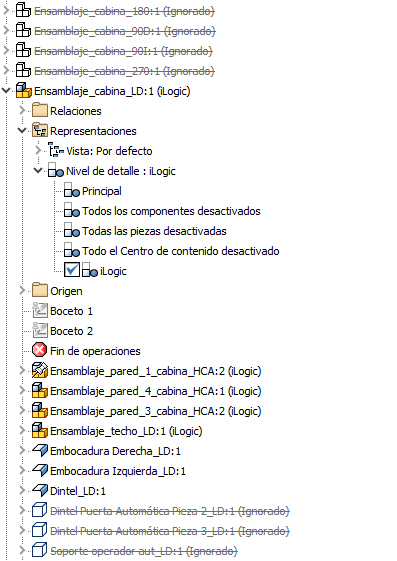


I change with a “Form” between different sub-assemblies with a rule, but it doesn’t change to the level of detail “ilogic” of the sub-assembly. It remains in the “master” level of detail:





and I have to change it manually:



If I change in the “Form” a new changed version of the model, I Will have to update and change manually this LOD again until I do it with everyone of them.

When I have changed al lof them and I save the Project, it runs properly in the model Windows. I can change differents parameters with the form and it recalculate fine. But If I open the drawings, and if I update them, when I save the drawings Inventor shows me a window where tells me that I have to save the master assembly “.iam” and somes sub-assemblies “.iam” too. If I click “Yes”, when I close the “.idw” and re-open the master “.iam”, I have again the master assembly changed, with all the level of details changed to the master LOD again…

How can I avoid it? Do you have any suggestion on the way I work?

Thanks you!

Note: I have done the same process in other Project, that it’s similar in components and dimensions to this one, and I didn´t have any error of this kind with this Project.

Note 2: I have desactivated the option of change the model when I’m in the drawing environment, so this is not the problem: