Clad That!

(if you dare)

-technology system basic guide-

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Contents

1 Clad That! (if you dare)

The class will guide attendees to understand the workflow between different Autodesk applications in order to approach the complex shapes typical of contemporary architecture as well as the benefits that Autodesk technology and services can bring.

The main set of Fusion 360 modeling techniques will be covered, and the produced material will be then used in Revit to create building components and a BIM model, allowing the designer to approach complex geometries and assign attributes and customized parameters where needed.

The creation of preconstruction documentation is then possible and demonstrates how to the BIM and Manufacturing worlds combined together can bring effective and consistent benefits to the Architecture and Construction industry. The class will drive participants to investigate and test the potential of the interaction between the 2 approaches. The aim is showing how a BIM model could be prepared for creating the construction documentation using the interaction between Fusion 360 and Revit, with a particular focus on complex architectural cladding modeling.

The main inspiration for the set of exercises described in this document is the stunning building designed by **Zaha Hadid** for the **Reinhold Messner Mountain Museum**. This is located in Plan de Corones, Italy, and it is composed of a concrete structure whose architectural space is "smoothed" and bounded with concrete panels cladding. The scope of this class is reproducing a similar building and explaining how architects and engineers can take advantage of Autodesk solutions to approach complex designs and make more informed decisions.

2 Learning Objectives

- Understand the combined use of Revit and Fusion 360
- Understand the main modeling techniques in Fusion 360
- Understand the potential of Fusion 360 for the architectural design
- Prepare preconstruction documents combining BIM and Manufacturing approaches
- Create and Manage complex geometries and unusual architectural shapes
- Understand the main set of applications involved on this kind of Conceptual Design
- Understand how to create BIM models for downstream uses

3 Scope of this document

The aim of the current document is outlining the use of the main set of applications (Fusion 360 and Revit) involved in the creation of complex architectural claddings.

This is organized across 8 exercises:

1. *Project Configuration*
2. *The Patch Environment – the First Branch*
3. *The Freeform Environment – one Branch and one Theatre*
4. *Solid Modeling – the Entrance*
5. *Terrain Creation*
6. *Fusion 360 to Revit*
7. *Drawing Production for Panel Assemblies*
8. *4D Sequencing for Panels Installation*

These exercises will show specific features in order to help the designers to better understand how to deal with this technology system while approaching a complex and contemporary architectural shapes and claddings.

4 Related BIM uses

According to **Penn State University (**https://www.bim.psu.edu/), a **BIM use** can be defined as:

> *"a method of applying Building Information Modeling during a facility's lifecycle to achieve one or more specific objectives."*

So, the **BIM use** defines the way an information model is going to be used to carry out specific activities and reach specific achievements.

The solution described in this document can be connected to the following uses.

Name	Description
Model Authoring	*The process to represent the design intent and all possible related scenarios by means of an information model.*
Drawing Production	*The process based on information models to support the creation of consistent drawings, annotations and schedules related to all the model components.*
3D Coordination	*The model-based process to find physical and by clearance interferences in a multidisciplinary integrated information model.*
4D Modeling	*The process for simulating time-based analysis, such as construction sequencing, in a model on building projects.*
Visualization	*The process based on information models allowing to visualize and represent the design with realistic 3D views, different projection types and styles.*

5 Your AU Expert

I am an architect based in Rome (Italy), with 17 years of experience in BIM.
I started working for Autodesk in 2005 as an Autodesk Authorized Consultant and finally I joined the Company in 2012 as a member of Autodesk Consulting, on the EMEA BIM Transformation Team. I am experienced on BIM transformation and implementations, Revit guru and architecture enthusiast.
My energies are spent on shaping an architectural oriented set of services, taking advantage of Autodesk applications with the BIM approach and adding a bit of "non-standard / non-linear" thinking.

When I had the opportunity to develop a framework to approach the Conceptual Design Process, I tried to forget the software features and try to understand how an architect thinks instead: several unusual and unexpected results came from this approach and so many new software capabilities can be used to face complex designs.

Although I precisely know that the final answer to life, universe and everything is 42, I am still working hard to find a way for improving and optimizing the conceptual design BIM approach.

During my spare time I still continue dealing with BIM (I'm an enthusiast, you know), playing basketball, exercise with my lightsaber, riding my Harley and strumming my ukulele although not necessarily in this order.

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6 The Messner Mountain Museum by Zaha Hadid

The Messner Mountain Museum has been taken as a reference for this document and the related exercises.

The museum has been designed by Zaha Hadid architect and according to the "Messner Mountain Museum" website (http://www.messner-mountain-museum.it/en/corones/museum/):

> *MMM Corones, situated on the summit plateau of Kronplatz (2,275 m), is dedicated to traditional mountaineering, which has strongly influenced – and been strongly influenced by – Reinhold Messner. The museum is devoted to mountain history and also offers unique views of the great mountain walls of the Dolomites and the Alps. At the edge of the most magnificent viewing platform in South Tyrol, in the distinctive museum building created by Zaha Hadid, the view goes beyond the borders of South Tyrol to all points of the compass: from the Lienz Dolomites in the east to the Ortler in the west, from the Marmolada in the south to the Zillertal Alps in the north.*

Reinhold Messner, the first man to climb Everest without supplemental oxygen, says about the museum:

> *MMM Corones on Kronplatz – between the Gader Valley, Olang and the Puster Valley – is the final act in the Messner Mountain Museum project (which comprises a total of six facilities). On the edge of South Tyrol's mountain plateau with the most spectacular views, in the unique museum architecture created by Zaha Hadid, I present the crowning of traditional mountaineering.*

Kronplatz offers views beyond the borders of South Tyrol to all points of the compass: from the Lienz Dolomites in the east to the Ortler in the west, from the Marmolada in the south to the Zillertal Alps in the north. The museum is a mirror of the world of my childhood - the Geislerspitzen, the central buttress of the Heiligkreuzkofel (the most difficult climb in my whole life) and the glaciated granite mountains of the Ahrn Valley. On Kronplatz I present the development of modern mountaineering and 250 years of progress with regard to the equipment. I speak of triumphs and tragedies on the world's most famous peaks – the Matterhorn, Cerro Torre, K2 – and the depiction of our activity, however contradictory it may seem. As in my other museums, I shed light on alpinism with the help of relics, thoughts, works of art (pictures and sculptures) and by reflecting the outside mountain backcloth in the interior of MMM Corones.

As the storyteller of traditional mountaineering, it is not my intention to judge or dramatise but simply to condense human experience of a world that is my world, of the 250-year-old contest between man and the mountain. The focus is not on sport and records but on people, on the key contributors to mountaineering, including philosophers and pioneers who had the courage to take the 'golden step' from the idea to the deed, disregarding the question "Why?"

"Corones" is the Ladin word for "crown", like "Krone" in German. And Kronplatz – South Tyrol's famous mountain for skiers and hikers and a perfect launchpad for hanggliders and paragliders – is now home to the crowning piece of my mountain museum project, a place of quiet where people can slow down and enjoy unforgettable views. It is a place of withdrawal that opens up the human senses for the above and beyond, where the mountains become an experiential space and a part of our culture. In mental flights beyond all summits, they are revealed anew.

The museum web site also reports:

> *The first structure built to parametric standards in South Tyrol therefore stands on a mountain peak. Hadid was known for her free-form architecture, based on digital design techniques. Nature and environment play a decisive role, with the architectural forms seeming to blend into the surroundings. This is why concrete was chosen as the material for the exterior and the interior cladding, as no other material lends itself so well to casting in so many forms. It also best fits the topic of rock, both as regards look and feel. The colour of the concrete and the building itself – for the most part built underground in order to intrude as little as possible upon the landscape and to avoid further construction on the peak – fit naturally into the surrounding mountain landscape.*

This museum gave the inspiration for the solution described with this document; the reason if related to its complexity and architectural quality and to the need of defining some consistent methods to use Autodesk applications to create a consistent process to allow all the involved designers to coordinate their models without losing the design freedom. The main aspect of the museum which is relevant for the process here faced, is related to the creation of concrete cladding systems. This is the central topic for the described solution and involves different Autodesk tools.

7 Useful Information before to start

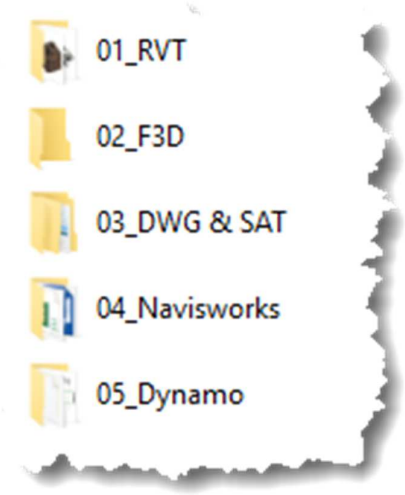
Just some information before starting the exercises, see following points.

7.1 Dataset

These exercises are supported by a dataset provided to all attendees.

The **Dataset** folder contains three sub-folders:

- **01_RVT** → Revit template, intermediate model and the final one;
- **02_F3D** → Fusion 360 files;
- **03_DWG & SAT** → AutoCAD and SAT exports from Fusion 360;
- **04_Navisworks** → Navisworks (NWF and NWC) files and the 4D inputs;
- **05_Dynamo** → the Dynamo Scripts and the related family templates;

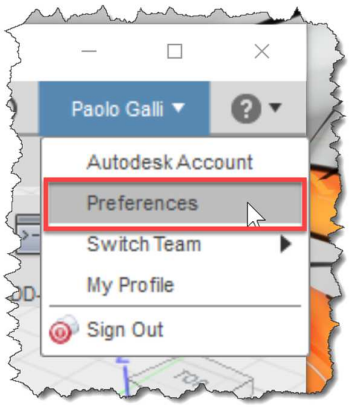


NOTE

- **Everyone has to create a folder in which the copy of the dataset has to be placed**
- **DO NOT work on the shared folder, otherwise there is the risk to prevent files to be used by other people**

7.2 Fusion 360: preparing the environment

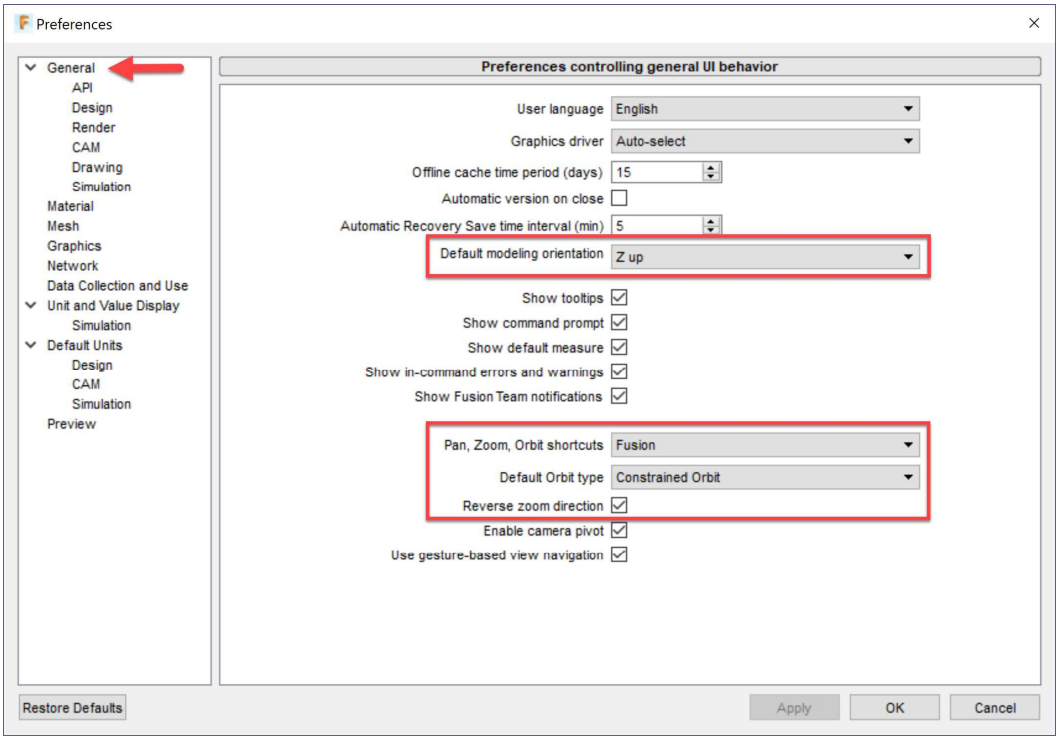
Fusion 360 is a software with a **mechanical design approach**, this means that there are a number of default settings which are oriented to the manufacturing standards. It is necessary to configure the environment so that this works in the way an AEC (Architecture, Engineering, Construction) designer is used to.



Click on the upper right corner, where the account user name is displayed and select “Preferences”.

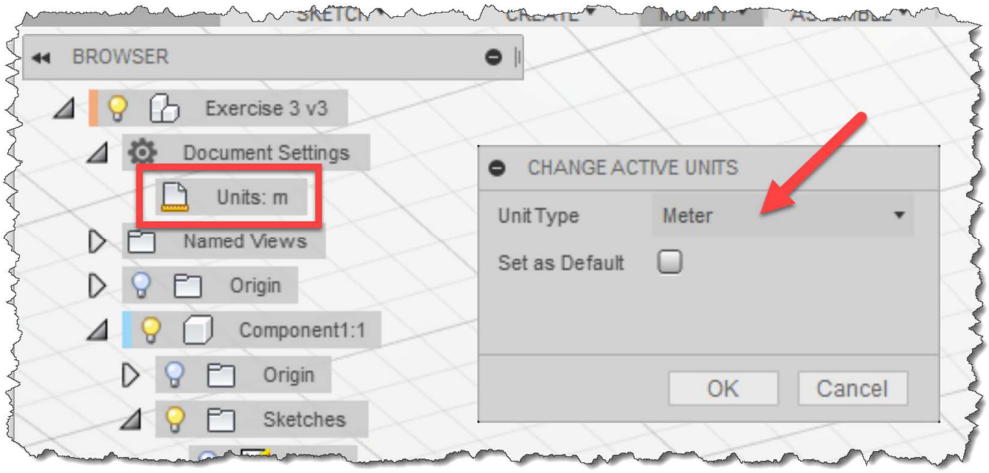
Here, setup the “General” section as shown on the next image, in order to:

- Define Z as the vertical axis (default Y);
- Check that “Fusion” is selected for “Pan, Zoom, Orbit shortcuts;
- Set the orbit type as “Constrained” (default “Free”);



Since Fusion 360 is a software oriented to manufacturing, the typical units used are millimeters. In some cases, depending on the size of the element to be modeled, the used

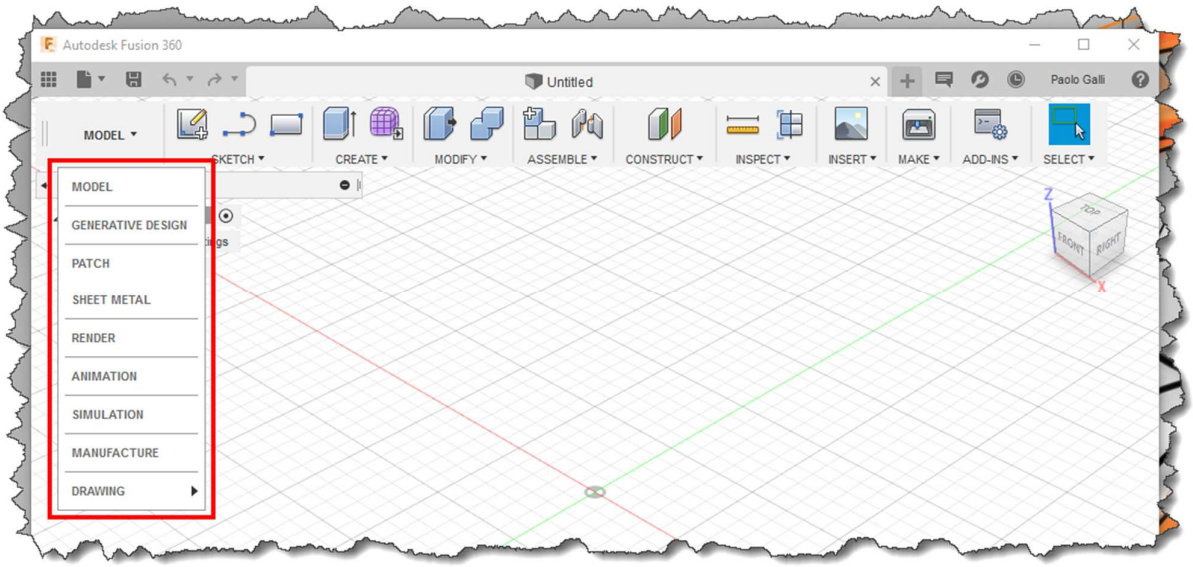
units could be meters; in this case, in the project tree under “Document Settings”, select “Units” and specify “Meter” for the “Unit Type” field.



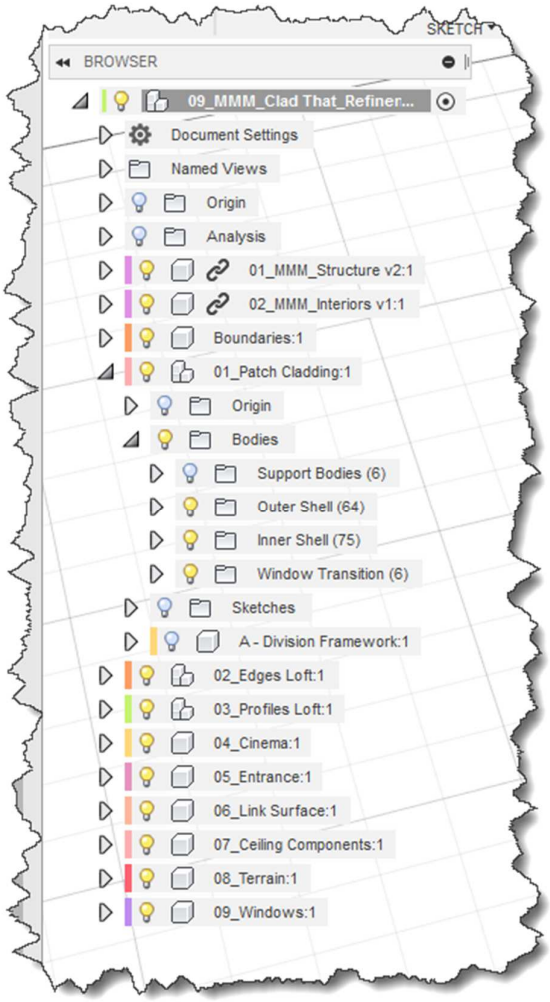
7.3 User Interface

Fusion 360 interface depends on the active environment; all commands are organized on the horizontal ribbon and they change according to the current work area.

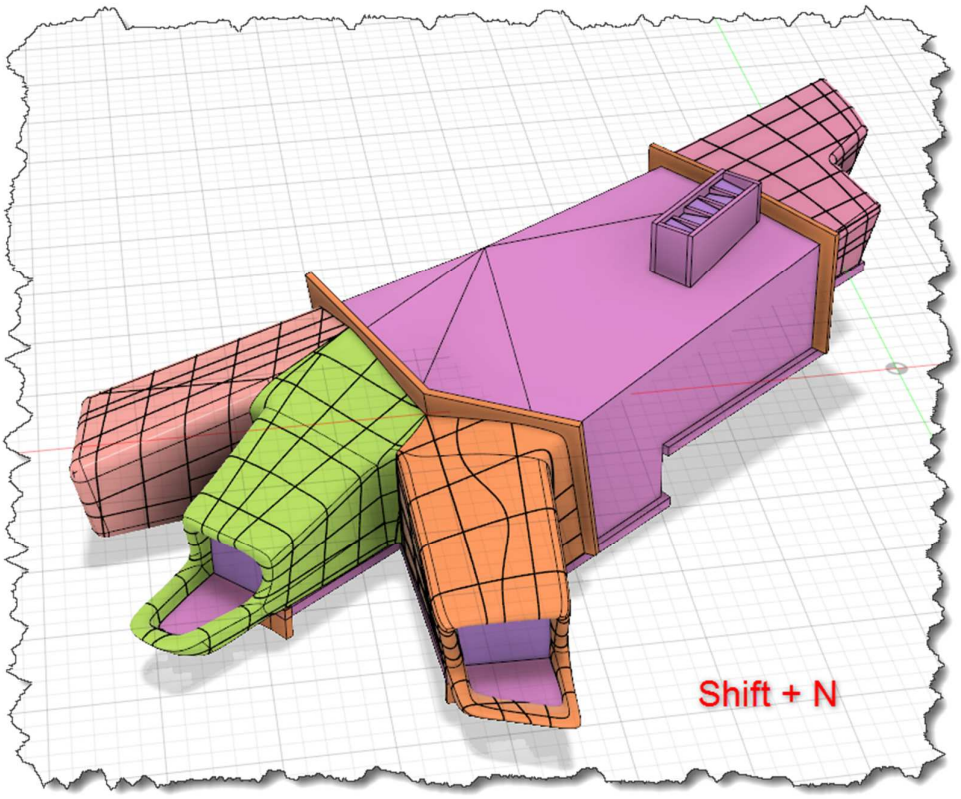
This can be accessed by clicking on the first button on the left of the ribbon. In this class, only the Model and Patch areas will be used.



The navigation tree is on the left of the user interface and this allows to navigate and organize the project components. Here it is possible to create, see and manage the components, bodies, sketches and other settings such as the project units.



This is a **tree organization** in which each component is assigned with a random color allowing the user to quickly identify what are the bodies belonging to the same component in the work area (press Shift + N to assign the color to each specific component on the work area).

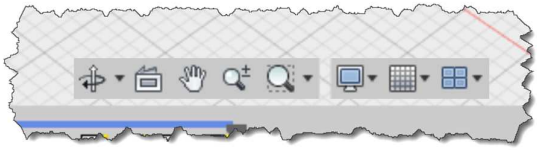


Next to each component name there is a triangle; by clicking on this the component content expands, and all the related elements can be viewed, turned on/off, renamed and deleted/removed.

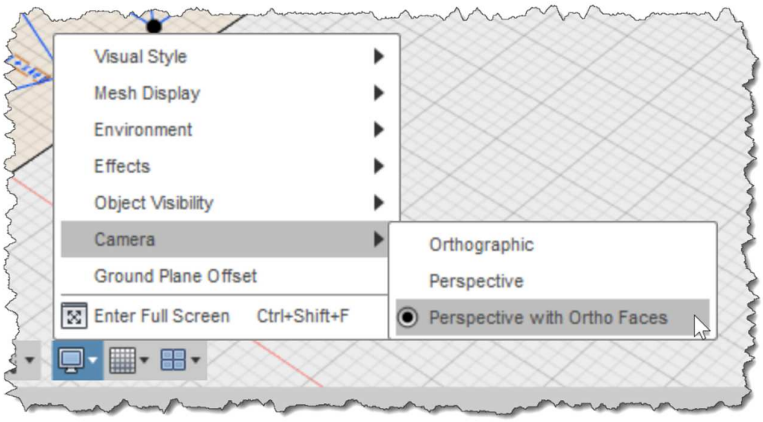
A **component** can usually consist of the following elements, which can be found below the component tree:

- **Origin**
- **Sketches**
- **Bodies**
- **Constructions** (reference points, lines, planes)

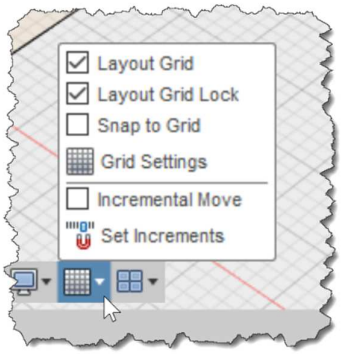
The lower part of the work area contains the visibility bar which allows to setup the visual styles and the viewports.



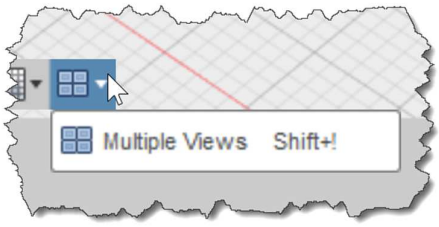
The first icon is for setting up the camera and the visual appearance of the model (effects such as shadows, reflections, ground plane…) and the camera properties: set this as "Perspective with Ortho Faces". This option allows to maintain the perspective unless a face of the view cube is selected; in this case the "elevation" view will be displayed using parallel projections.



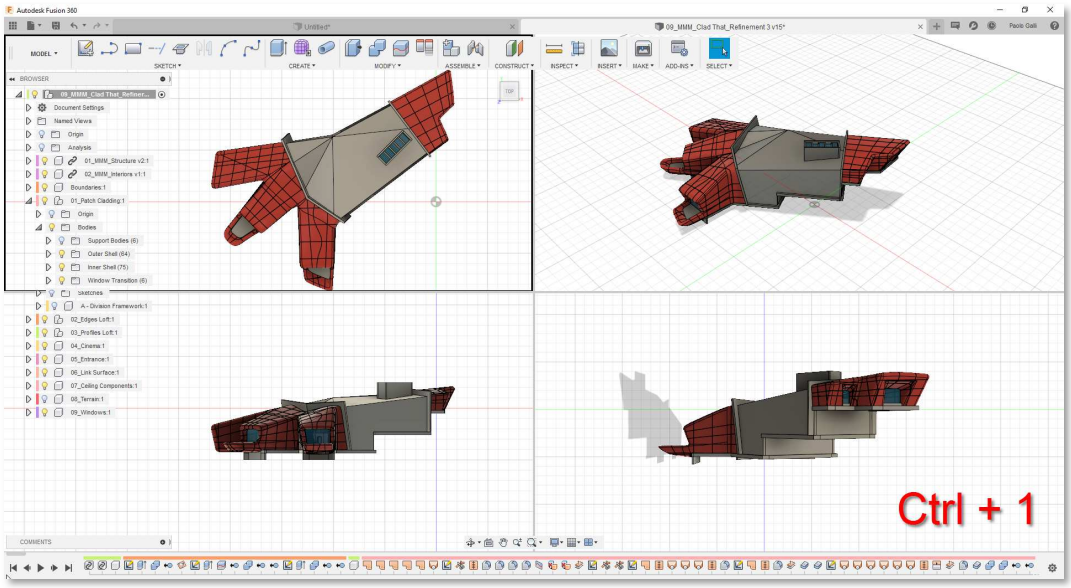
The second icon is for showing/hiding the grid and to switch on/off the snap to grid.



The last button allows to split the work area in multiple viewports or maintain the single viewport provided by default.

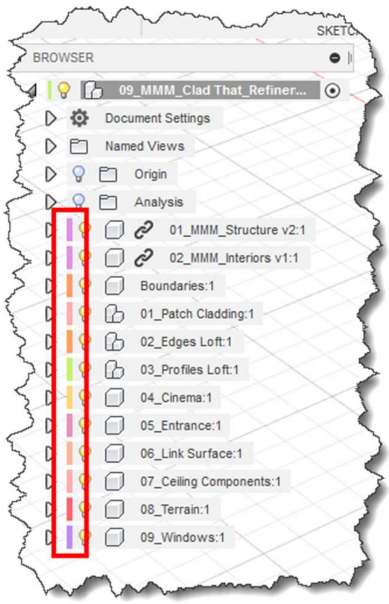


When multiple views are shown, it is also possible to synchronize them in order to have the alignment between parallel projections maintained while zooming or panning.



The lowest part of the user interface hosts the timeline which reports all the events occurred in the model, in terms of used commands and the components to whom those command have been applied.

Association between commands and components is reflected by the color bar above the commands list, as shown in the previous picture; this assumes the same color displayed next to each component in the project tree (image on the right).

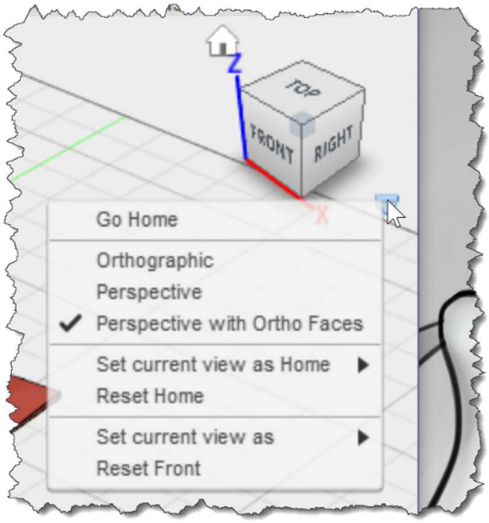


7.4 Navigation on Fusion 360

The navigation, once the initial setup has been done, it is pretty much the same to that in Revit:

- Scroll the mouse wheel to zoom in and out
- Double click on the mouse wheel to zoom to fit
- Press and drag the mouse wheel to pan
- Press shift and the mouse wheel to orbit

There is a view cube which gives the ability to quickly rotate the point of view according to the cube faces and also gives other options such as switching from perspective to orthographic view.



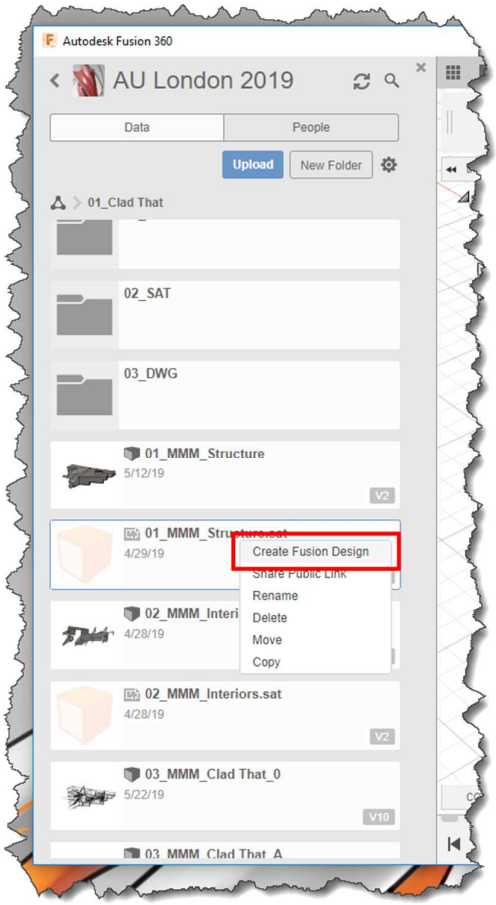
NOTE
The use of a 3D mouse is suggested since it enhances the navigation experience and efficiency in Fusion 360.

7.5 File formats

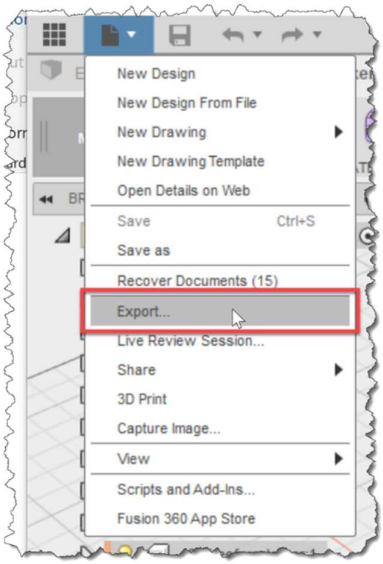
The native Fusion 360 file format is F3D but it can use a number of other file formats.

Since the file repository is a A360 project and Fusion is based on cloud shared data, the use of external files requires the conversion to “Fusion Design”.

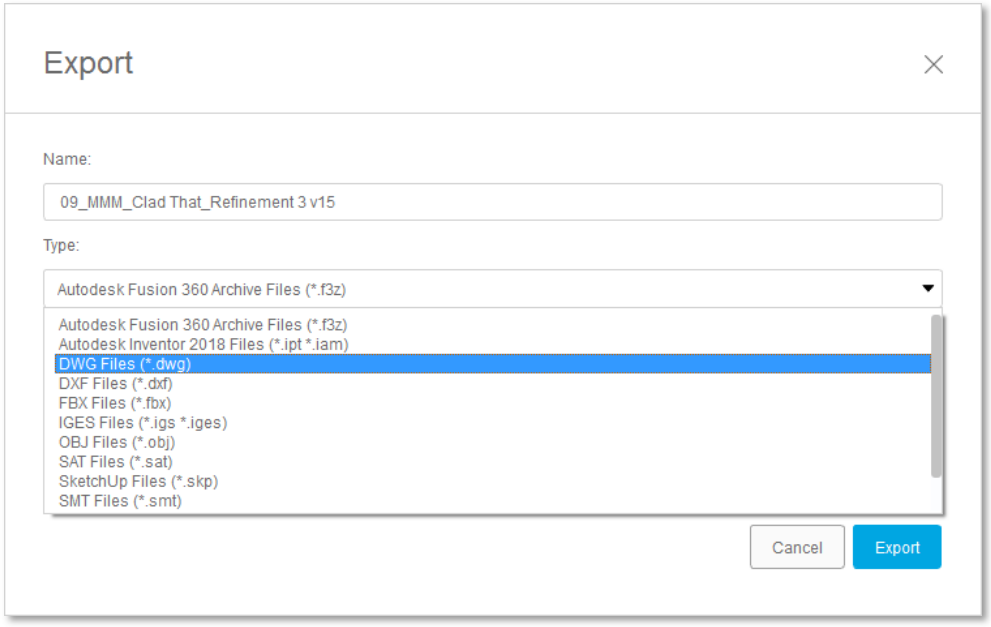
After a file has been uploaded in A360, it is necessary to start the conversion to “Fusion Design” in order to have the material prepared to be used, see image below. This is also required in the case of native F3D files.



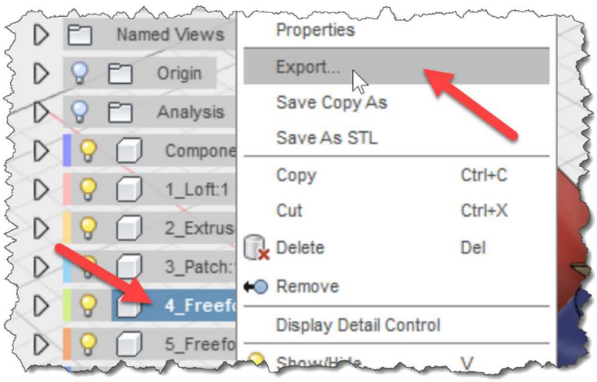
It is possible to export the project material to other file formats, too. User can choose if exporting the entire project or a single component. In the first case, the “export” command is located on the main menu, as shown in the next picture.



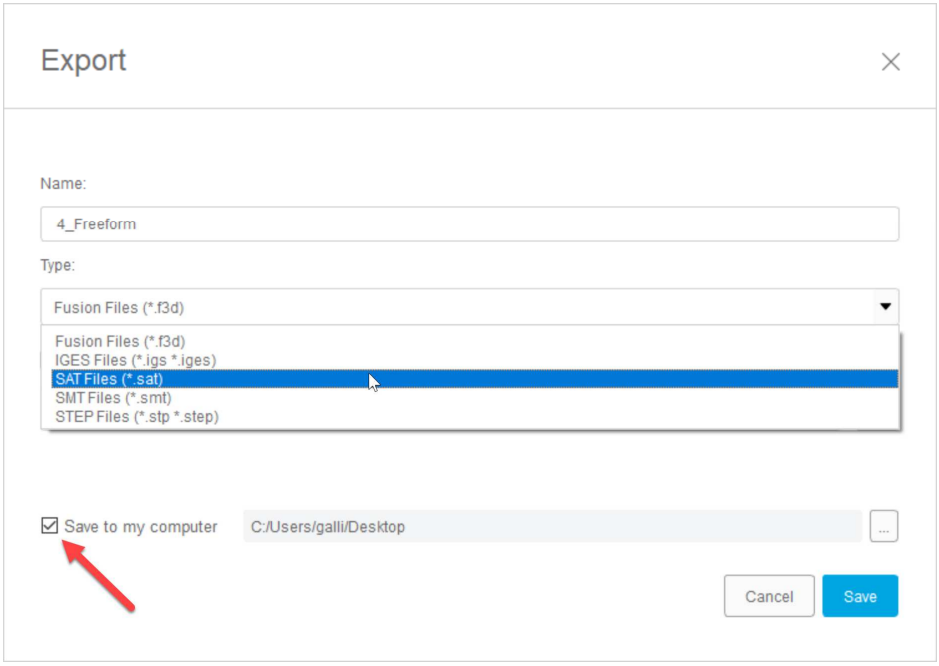
Fusion 360 allows to export the entire project in multiple file formats using the command shown before.



If a single component has to be exported, it is needed to identify the component in the project tree and then right click on it to find the “Export…” command.

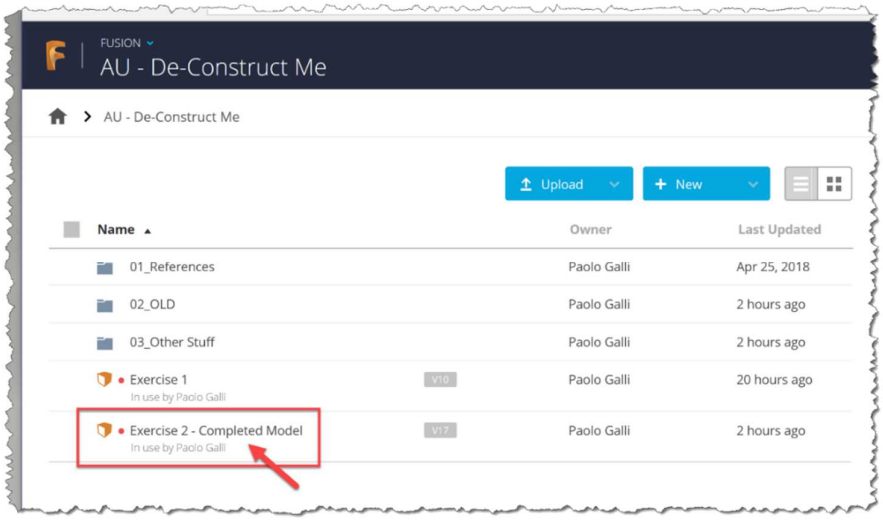


Fusion 360 allows to choose whether exporting files on the cloud service or locally.

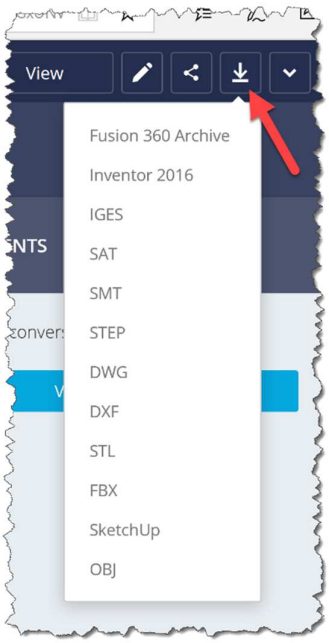


Fusion files can be also saved in a number of other formats using A360 in the web browser:

1. Identify the project, then the file and click on it.



2. Once file has been selected, a new page is displayed and there, click on the *down arrow* located on the upper right corner of the screen. This allows to export the entire project to several different formats, including Inventor 2016 and DWG.

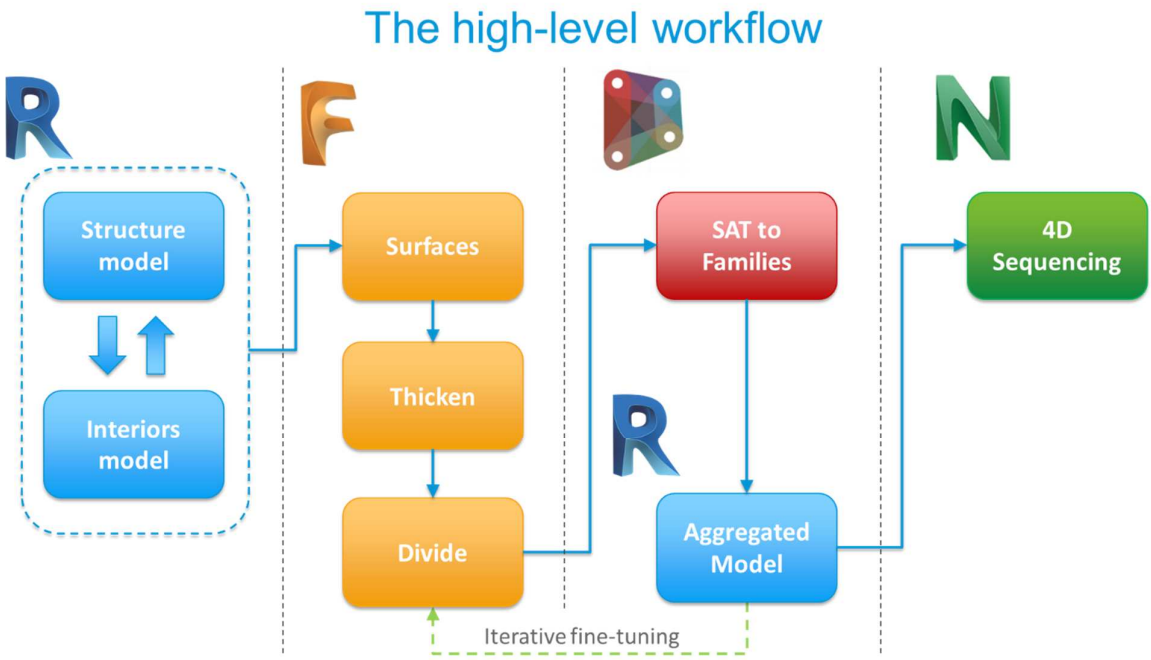


7.6 Autodesk Account

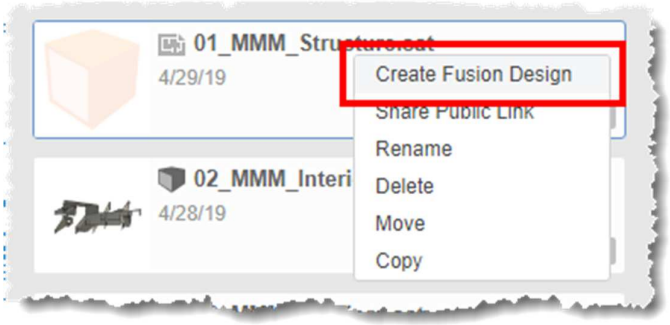
Users must sign-in the Autodesk Account in order to use the A360 cloud folder and Fusion 360.

8 High level process

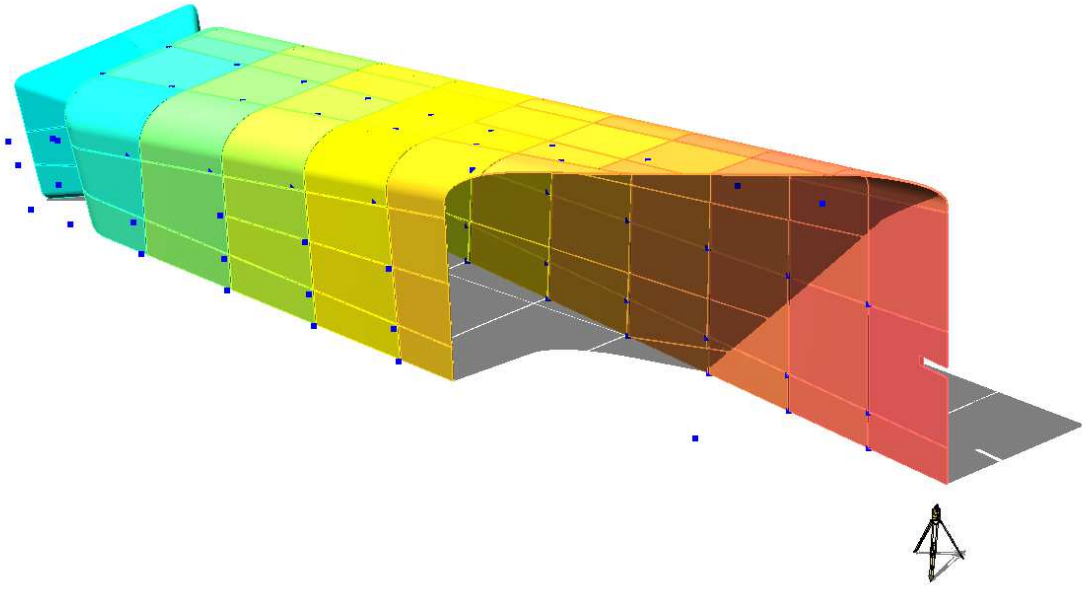
Following scheme shows the high-level process followed for this class.



- A Revit model is created in order to feed the cladding definition process. The Model can be composed of structural and architectural files, coordinated in terms of positioning, orientation, elevation as well as in terms of "copied and monitored" elements (reference to the Copy/Monitor command in Revit);
- Revit models are exported to SAT and uploaded to a specific A360 cloud project (myhub.autodesk360.com);
- Revit models are converted to Fusion Designs;



- Converted models are inserted as links in an empty Fusion 360 project so that updated files can be reloaded and there is no risk to modify for mistake the converted Revit files;
- Fusion is used to create the cladding surfaces, give them a thickness and divide the created components in panels according to the architectural design;
- The created components are then exported as SAT files;
- SAT files are imported in Revit using a Dynamo script which recognizes all the components inside the file as single editable families;
- Revit is then used to aggregate different models coming from Fusion 360 and to produce drawings;
- A dynamo script is used to define the installation sequence and write the related information on specific parameters;
- Revit is then used to create schedules, based on the parameters populated in the previous point, to feed Navisworks and simulate the panels installation sequence;

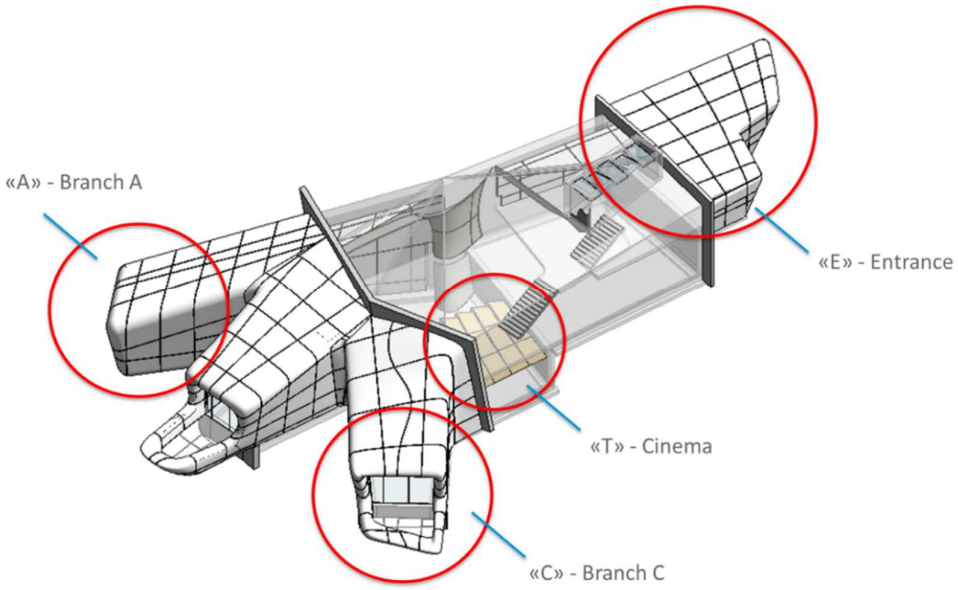


This process can be **iterative**: it is possible to bring back elements from Revit to Fusion in order to use them as a reference to refine the model.

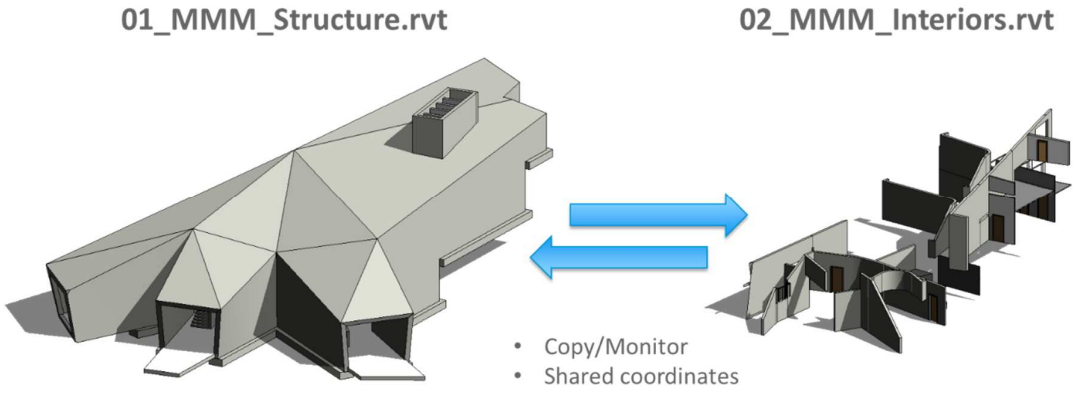
9 Exercise #1: Project Configuration

The scope of the workflow documented in this document is showing how a coordinated set of Revit files can be used to drive the modeling of complex architectural shapes in Fusion 360. For this reason, two Revit files have been provided with the dataset. These have been coordinated in terms of positioning and of copied / monitored elements (levels only, in this case).

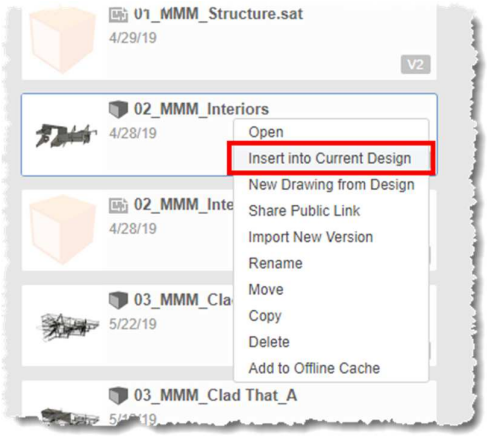
The model represents the structural concrete walls and the main interior walls of the building. Fusion is used to define the architectural surfaces. In order to understand the different parts of the building which will be used for the proposed exercises in this document, refer to the following image.



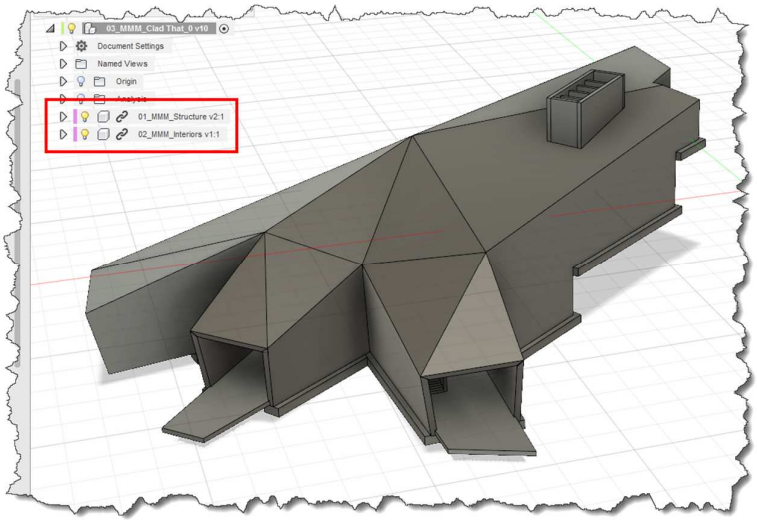
The Branch B modeling is not part of this current document.



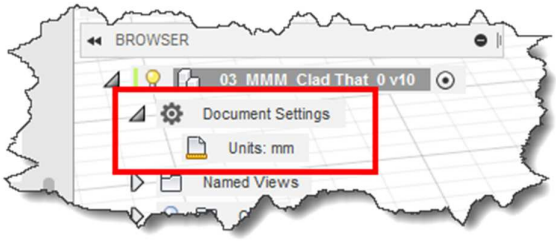
1. Upload the files on the A360 project folder;
2. Right click on them and “Create Fusion Design”;
3. Load both of the created Fusion files on an empty file – note that the file must be saved before loading external references;



4. The symbol of a chain appears close to the name of both the loaded models;



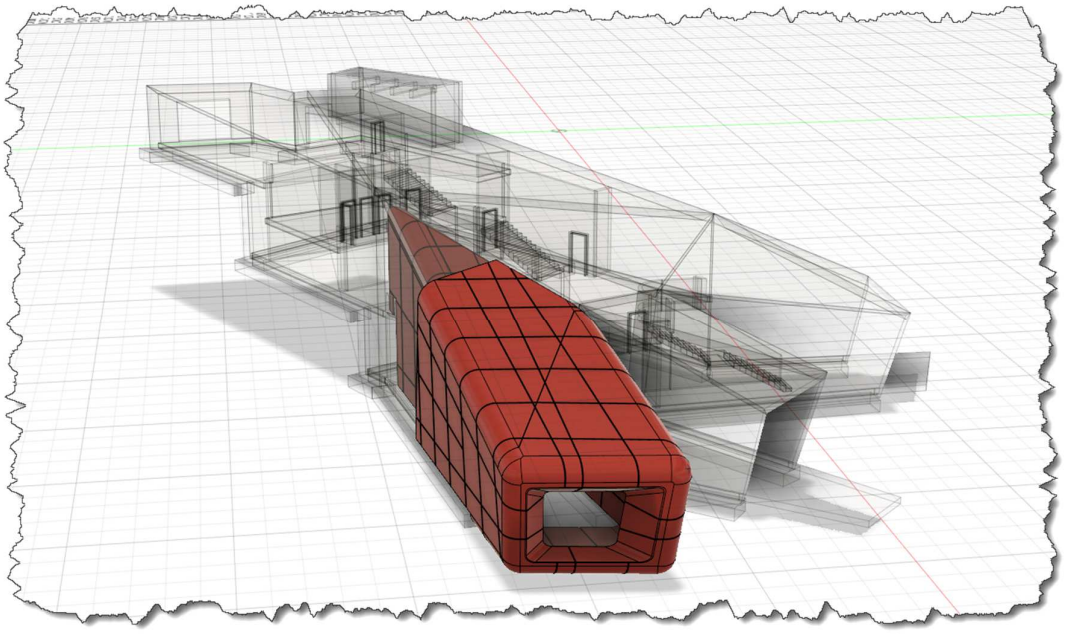
5. Check the units; they must remain in millimeters for this project;



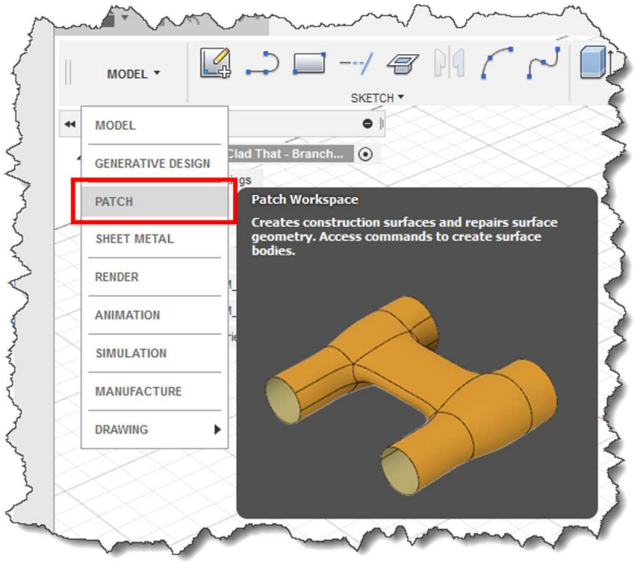
6. Save the file.

10 Exercise #2: The Patch Environment – the First Branch

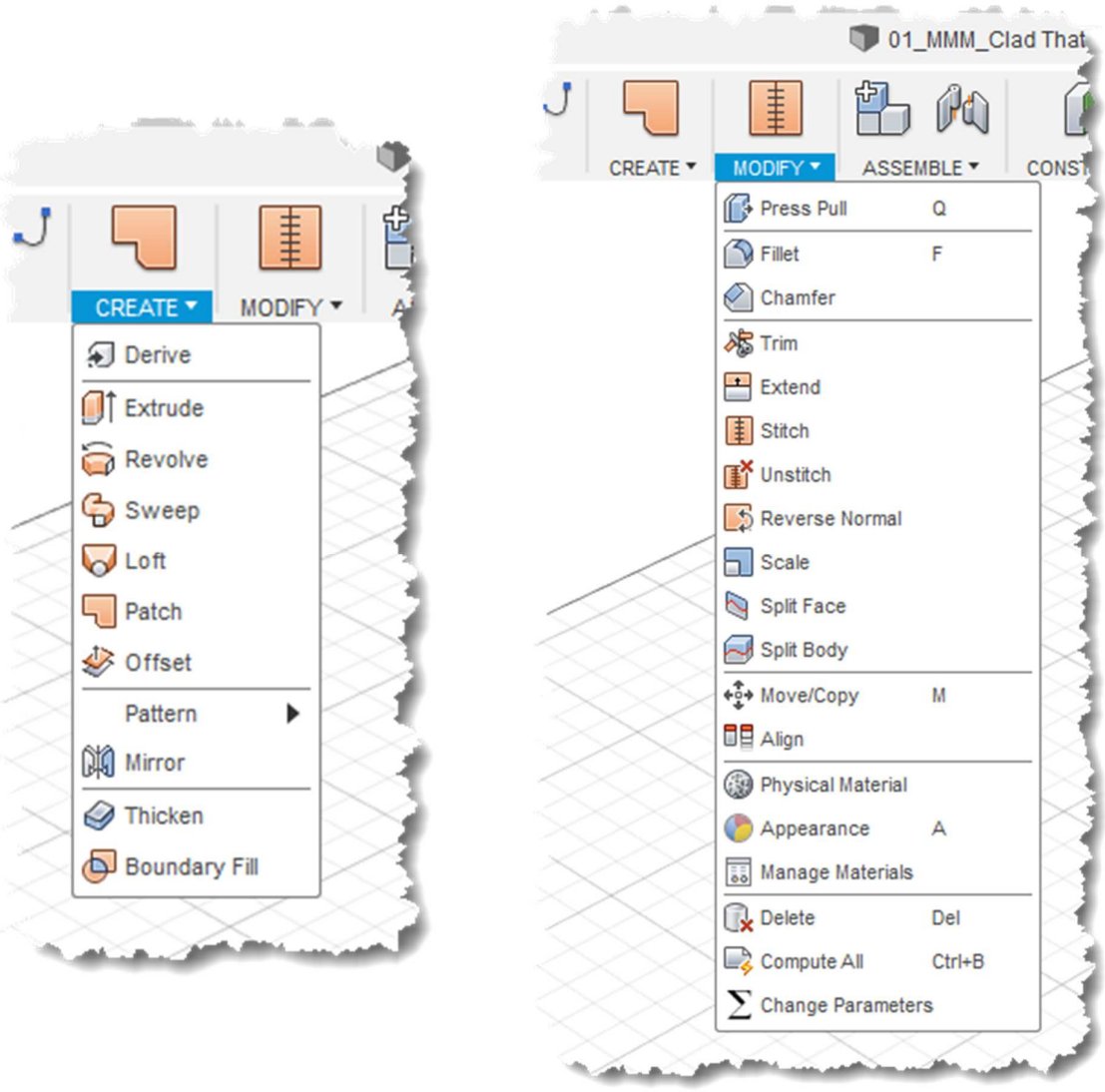
This exercise will cover the topics related to the creation of the cladding of the Branch A. Here the Patch environment of Fusion 360 will be used to investigate the capabilities of this set of features to create architectural finishes.



The Patch environment is the area of Fusion 360 which collects all the commands and features related to the creation and management of B-Rep surfaces. B-Rep stands for "Boundary Representation" and refers to surfaces and solid models which are defined by their boundary conditions (faces, edges and vertices).



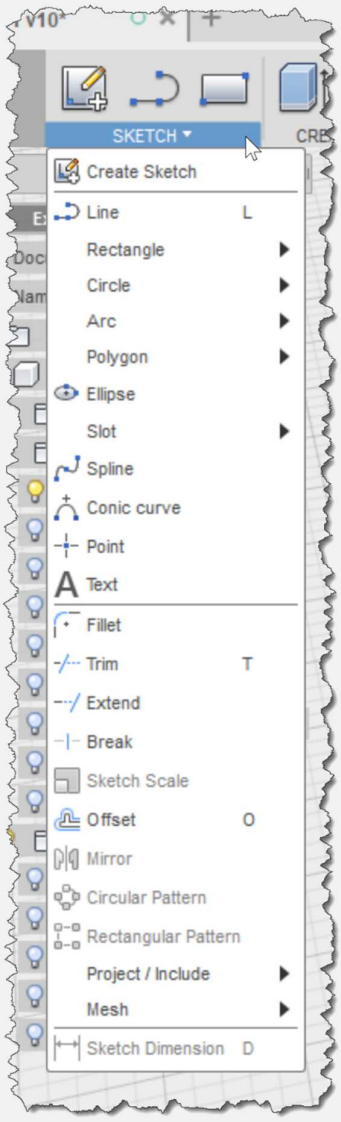
When this area is activated, the toolbar changes and displays the set of commands related to this workspace. Next picture shows the set of modeling and editing commands related to this environment.



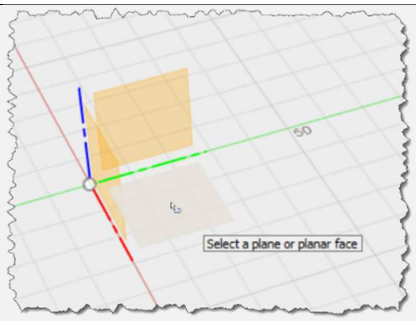
10.1 Sketching!

The majority of elements in Fusion 360, requires one or more sketches to be defined.
A sketch is a combination of unidimensional (1D) geometric primitives which can be defined and constrained by means of constrains and dimensions.
To create a sketch a reference plane has to be selected as the "container" of the sketch itself, but sketches can also exit from the plane and configure so a 3D sketch.

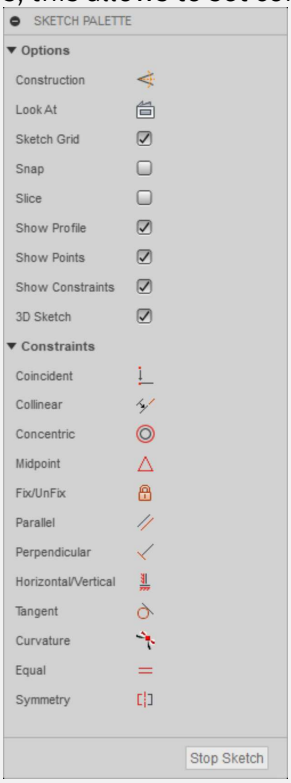
1. Click on any of the sketch elements in the sketch panel



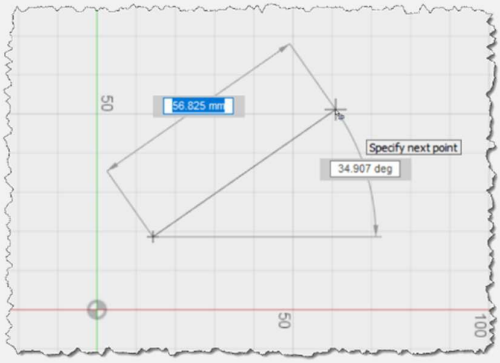
2. Fusion requires the selection of a reference plane to start the sketch; by default, the origin axis and planes (XY, XZ, YZ) are shown, but also other planes can be selected, even if those belong to 3D bodies;



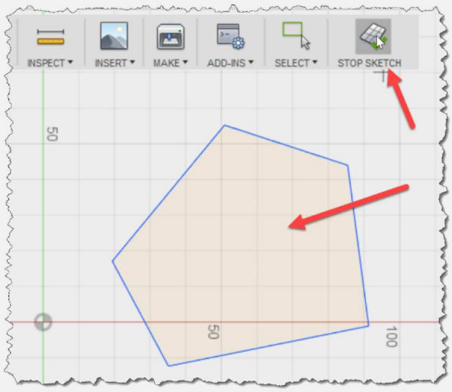
3. Fusion 360 orients the point of view to the selected plane;
4. The “Sketch Palette” appears; this allows to set constraints and other configurations;



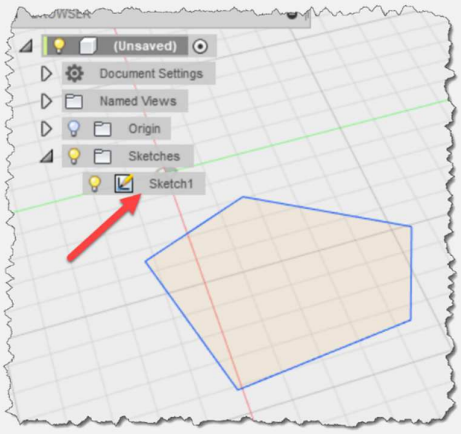
5. Click on the first point of the sketch; Fusion displays contextual information such as length and angle;



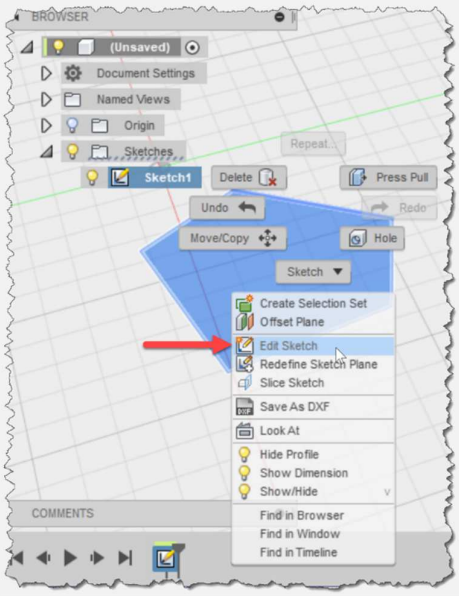
6. When the sketch is completed, click on “STOP SKETCH” on the upper right corner of the user interface. If the sketch is closed, this is automatically filled with a light orange solid pattern;



7. The new sketch now appears in the project tree where it can be renamed; the sketch is ready to be used;

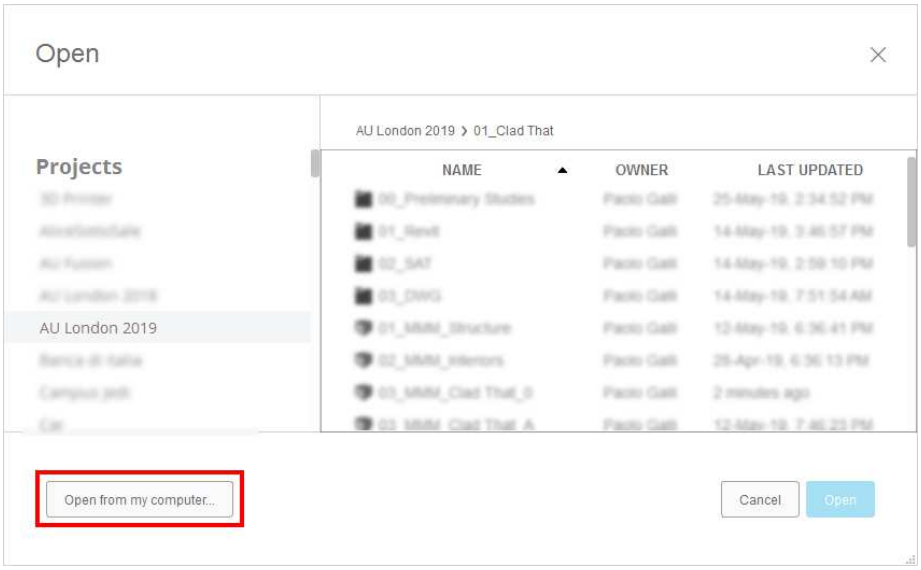


8. If it is necessary to edit this element, right click on either the project tree or directly on the sketch and click on “Edit”; this enters the sketch edit mode, displays the “Sketch Palette” and temporarily reverts the timeline to the point at which the sketch has been created.

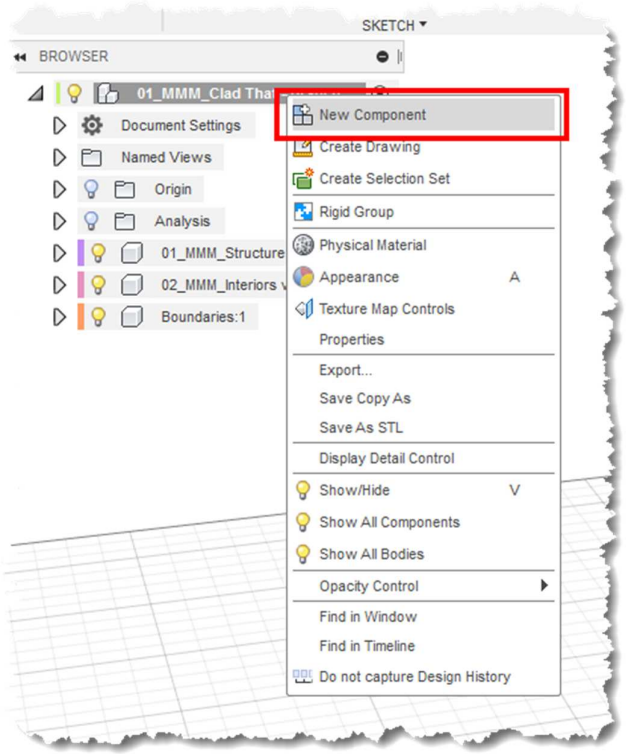


10.2 Base Geometry Creation

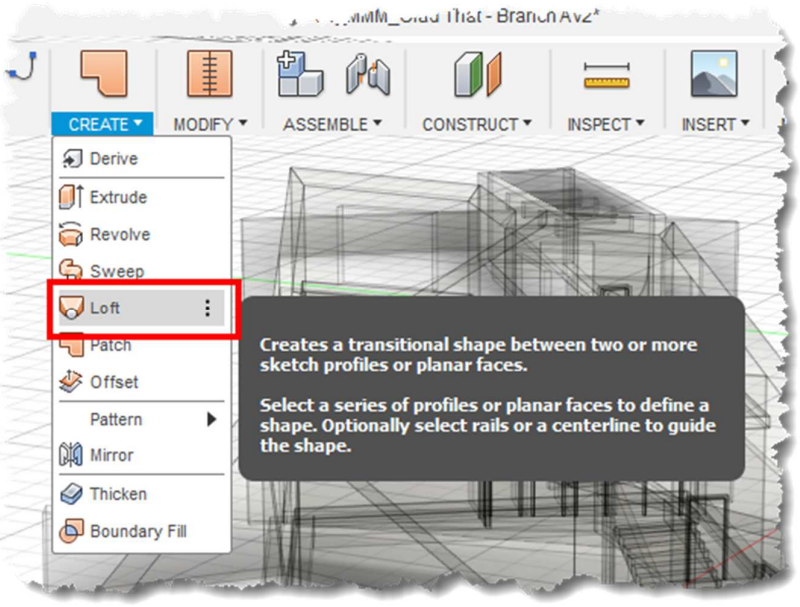
1. Open the file “01_MMM_Clad That – Branch A” or create a new drawing by opening the provided F3D file



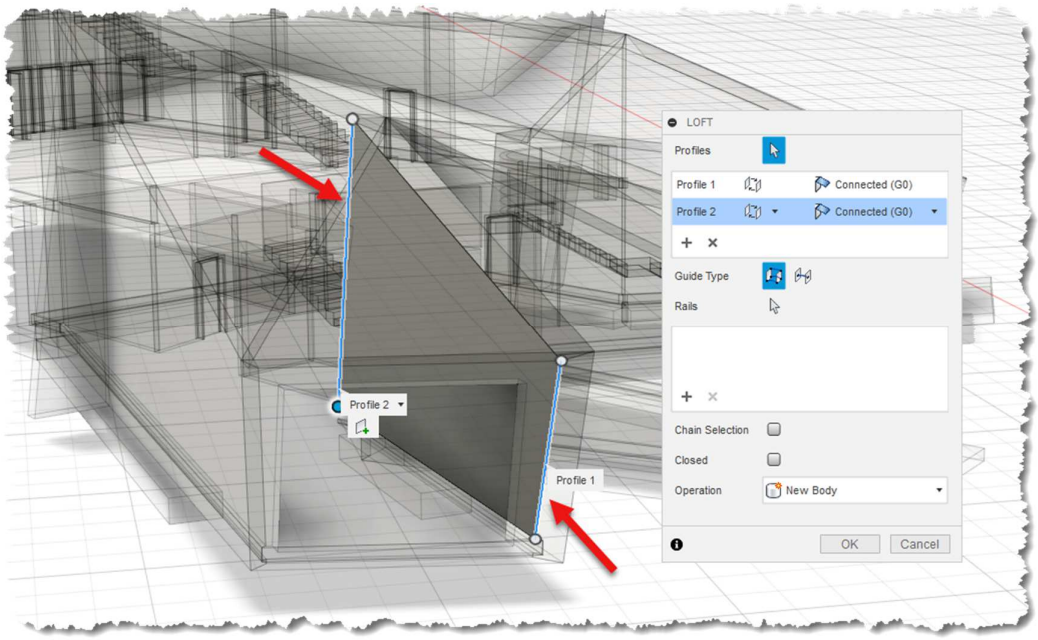
2. Right click on the file name at the top of the project tree and create a new component. Components are the functional units defined in Fusion; these are composed of Bodies, Sketches, Constructions. Name “03_Patch – A” the new created component;



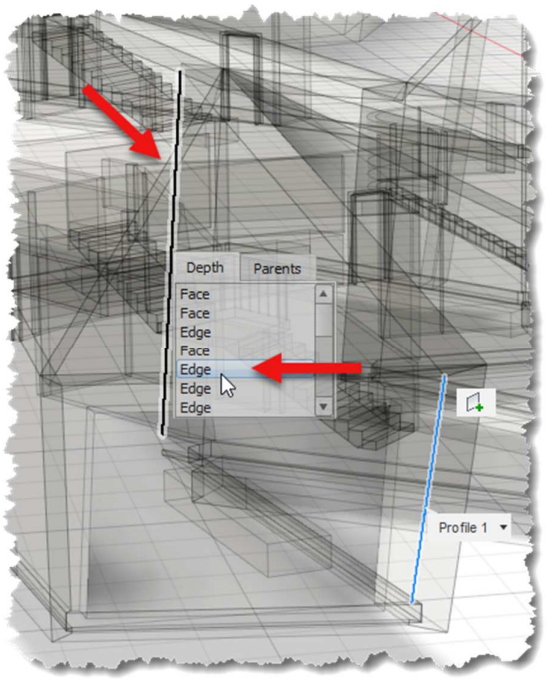
3. The component is automatically activated; for this reason the others are now displayed as transparent and in halftones. A black dot appears on the right of the active component name;
4. Click on the “Loft” command under the “Create” group;



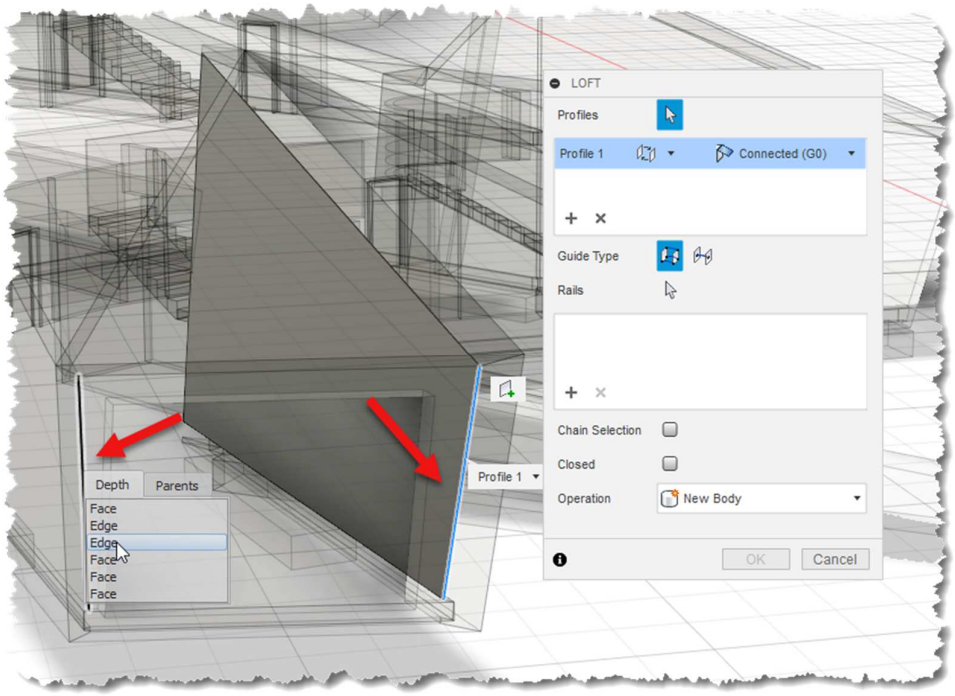
5. This command requires the selection of two or more profiles (they can be either closed or open) to create an interpolated surface across them; In this case the edges of the concrete walls will be used;



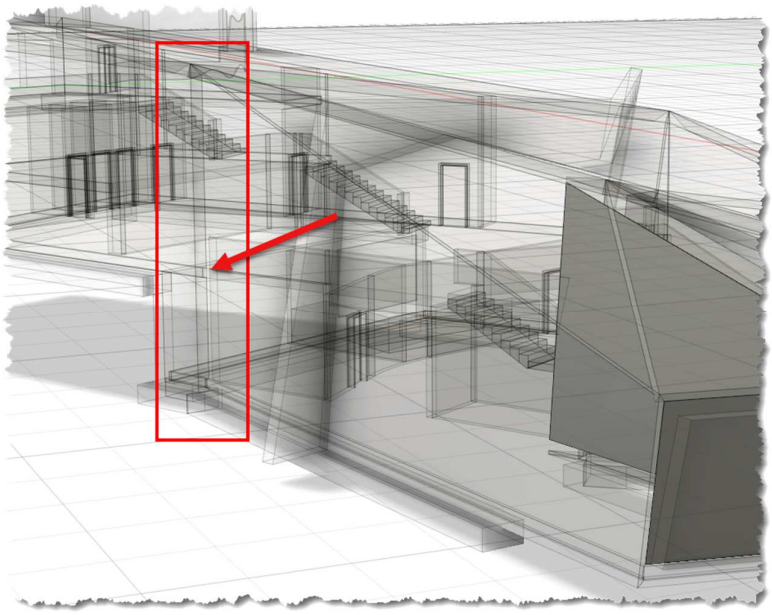
6. If an edge is not selectable, this means that it may behind another element. In this case, keep the left mouse button pressed and Fusion 360 will show a dialog listing all the elements below the mouse arrow that can be used as a reference;



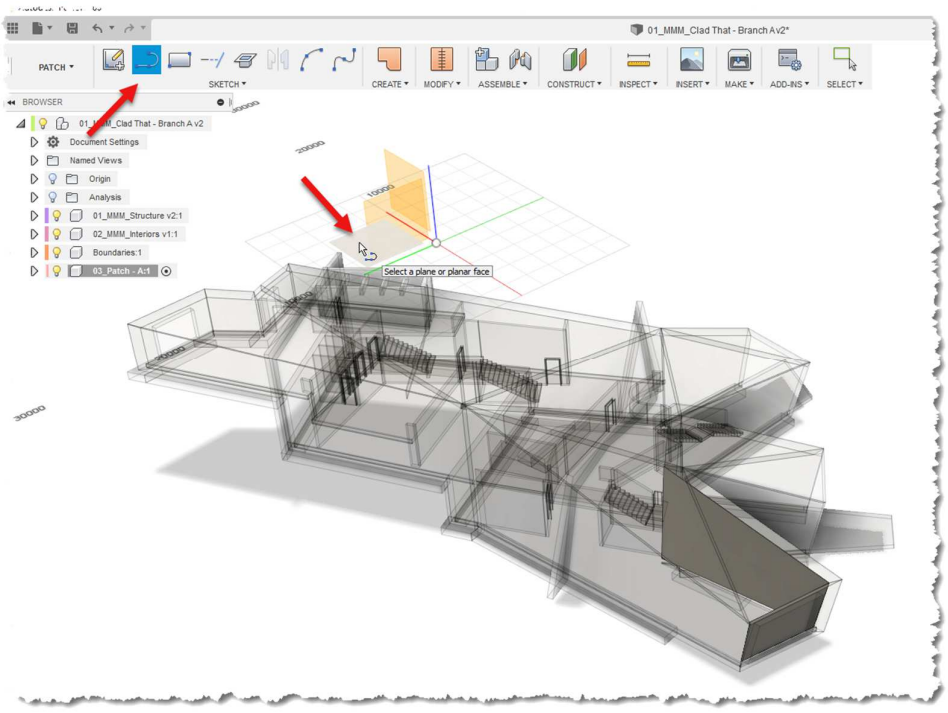
7. Click “Ok” when done and the first surface appears;
8. Repeat the same sequence for the shorter side (that one with the opening) of the Branch A and click “Ok” to create the related surface;



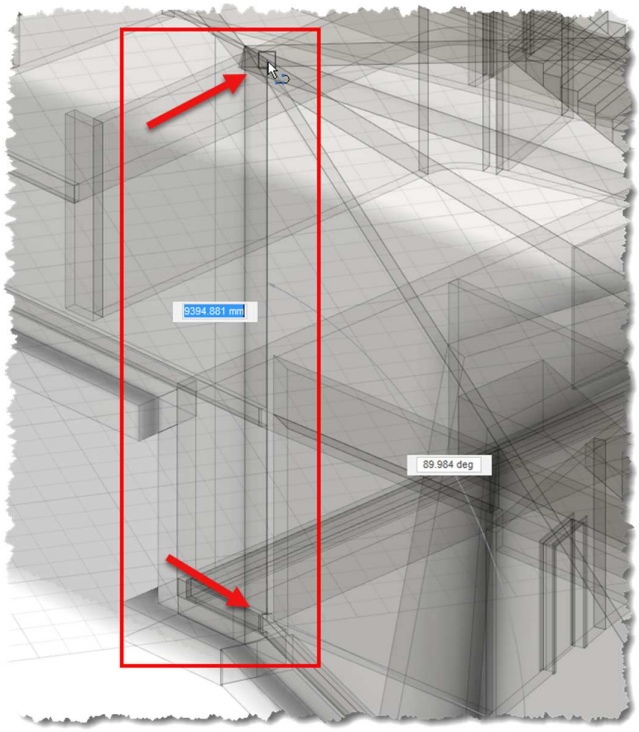
9. The third face cannot be modeled using the wall edges since a floor breaks in two parts the farther edge, as shown on the following picture;



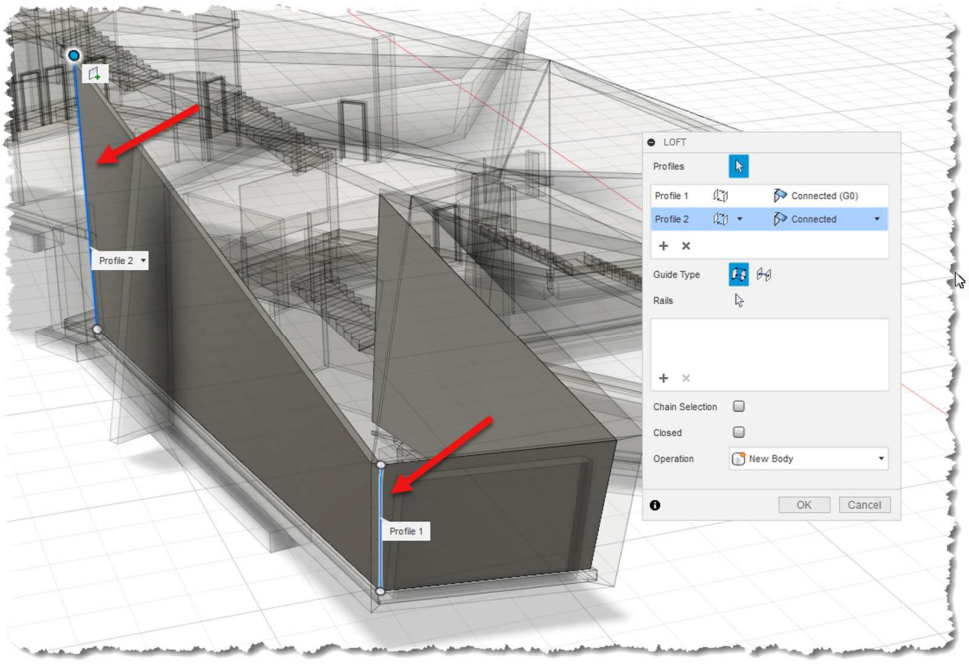
10. A sketch must be created to replicate the continuous geometry of the edge and use it to define a loft surface;
11. Create a new sketch by pressing the “Line” command and then select the XY plane as reference; in this case, although the XY plane is the “host” of the sketch, 3D elements outside the plane will be taken in count as drivers for the sketch geometry;



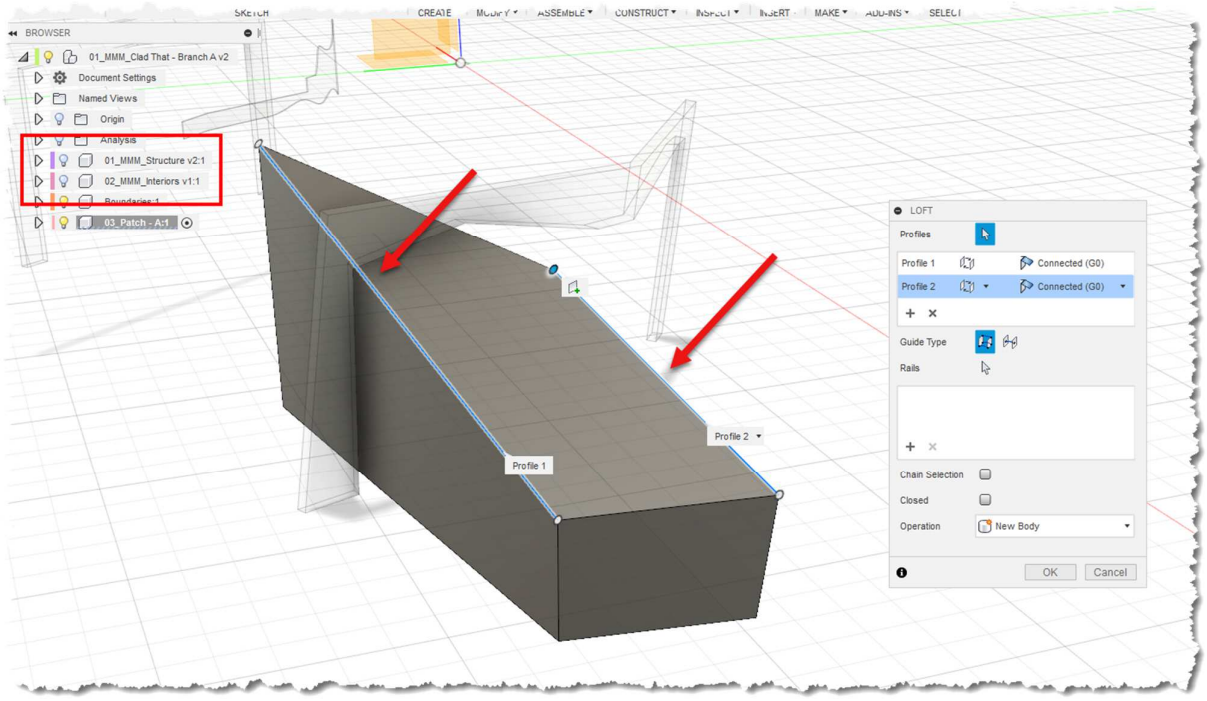
12. Fusion allows to snap on vertices even though they are outside the reference plane; click on the vertices representing the start and the end points of the wall edge;



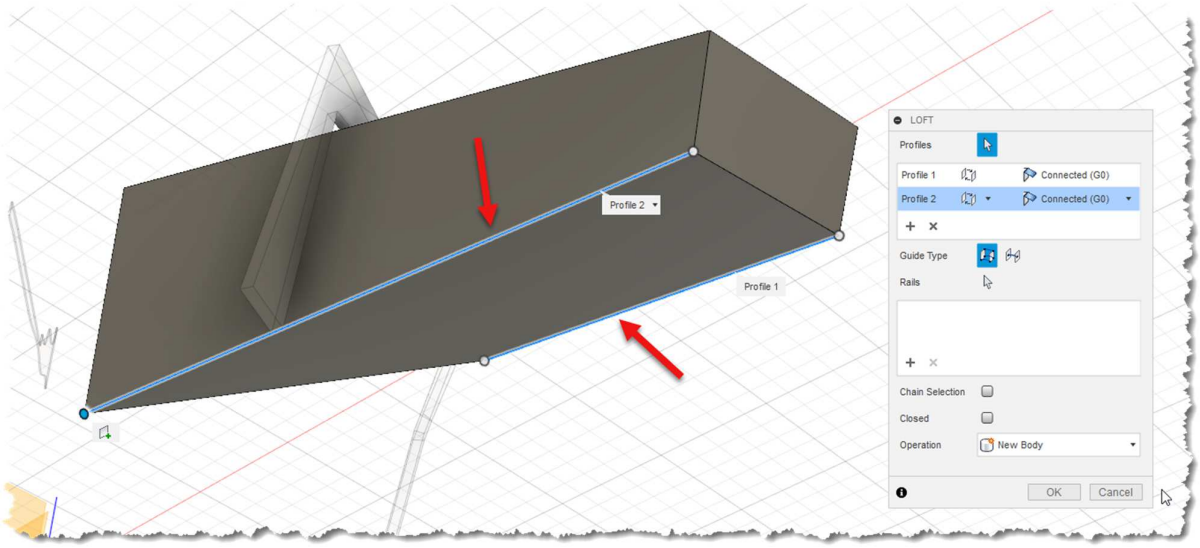
13. Click on “Stop Sketch” (on upper right corner) when done;
14. Create a loft surface using the wall edge close to the wall with the opening and the sketch line just created;



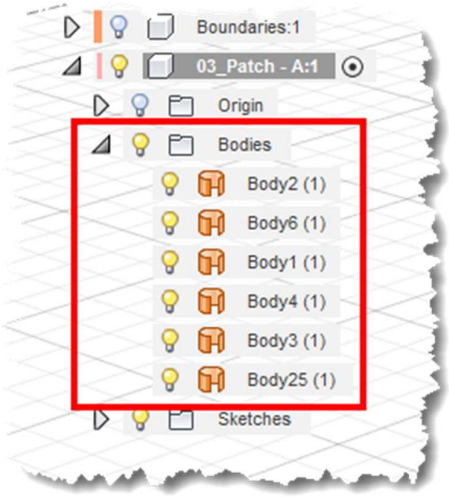
15. Turn off the visibility of the structural file and of the interior model;
16. Create a loft surface using as a reference the two top lines of the vertical faces as show on the following image;



17. Repeat the same command for creating the bottom surface; this will be used for the external cladding while it will be erased for the case of the interior finishing. Finalize the process by creating also the face on the rear side;



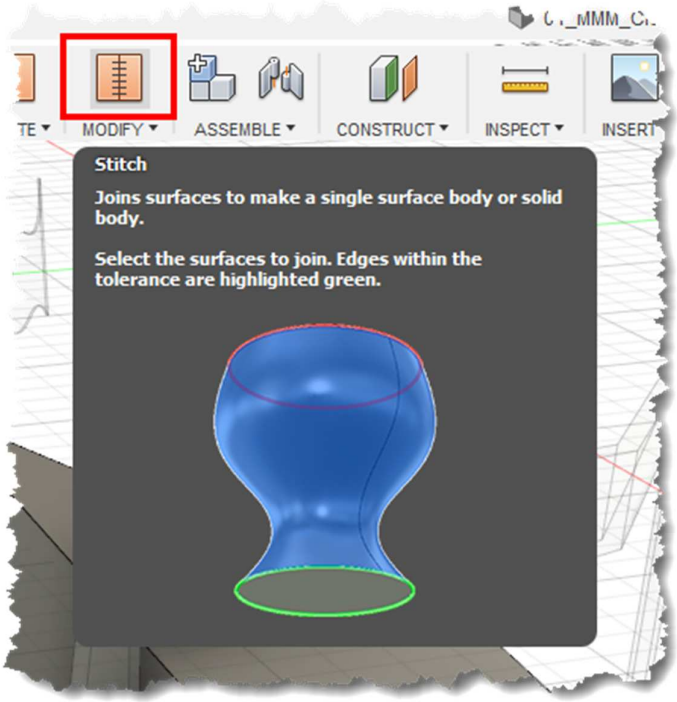
18. The project browser shows now six different bodies;



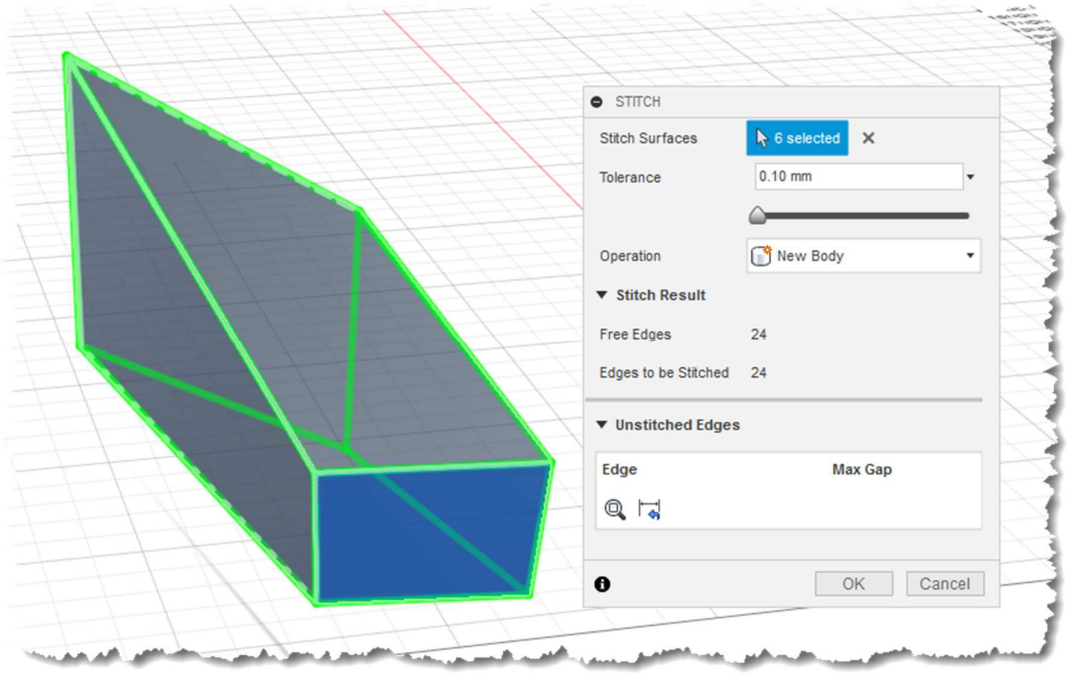
19. These six bodies are going to be used to create the external and internal shells; for this reason, they will be both joined and separated when this is needed in order to create the right offset geometries;

10.3 External and Internal Shells

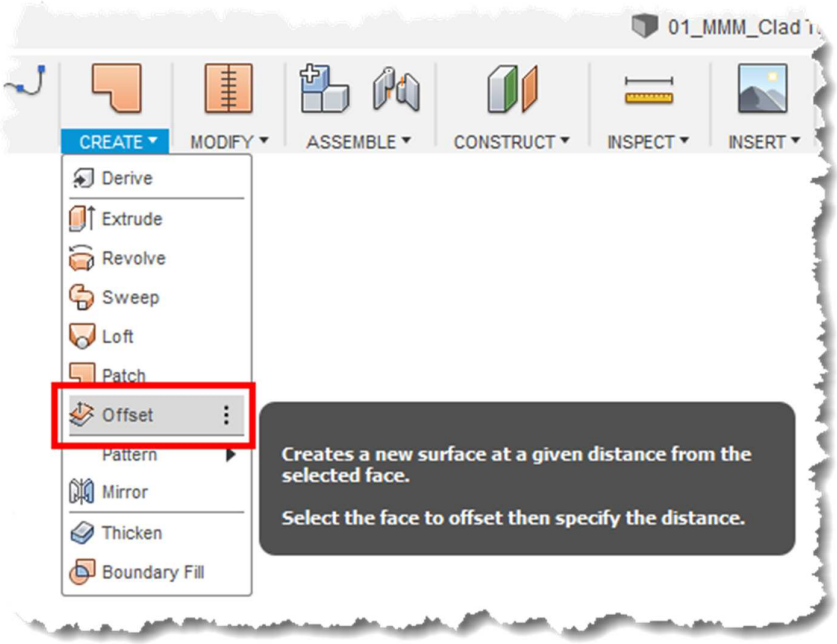
1. Use the “Stitch” command to join the six surfaces into a single body;



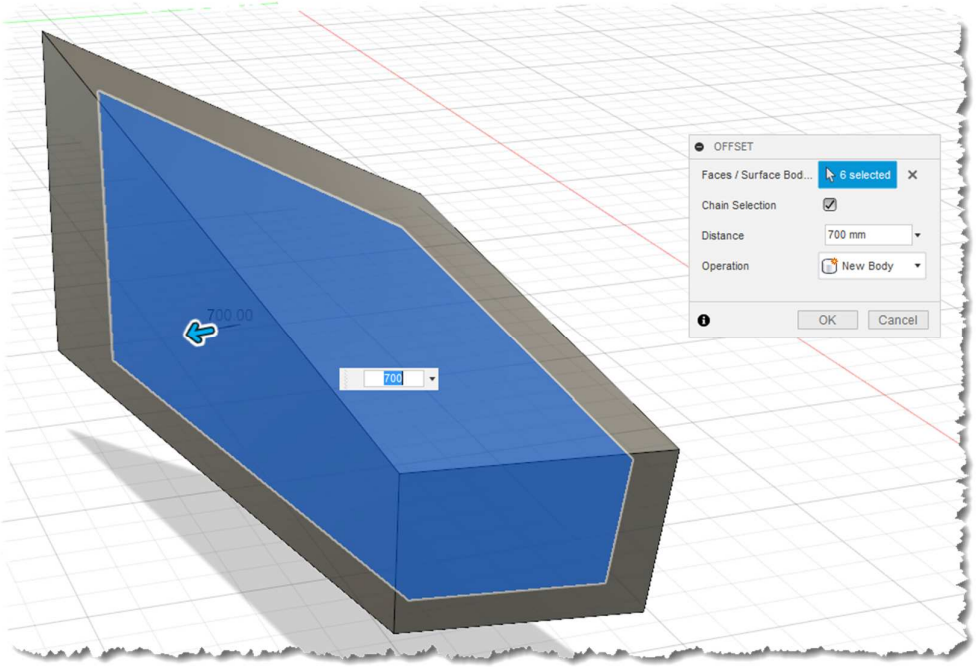
2. Select the six surfaces and click “Ok”;



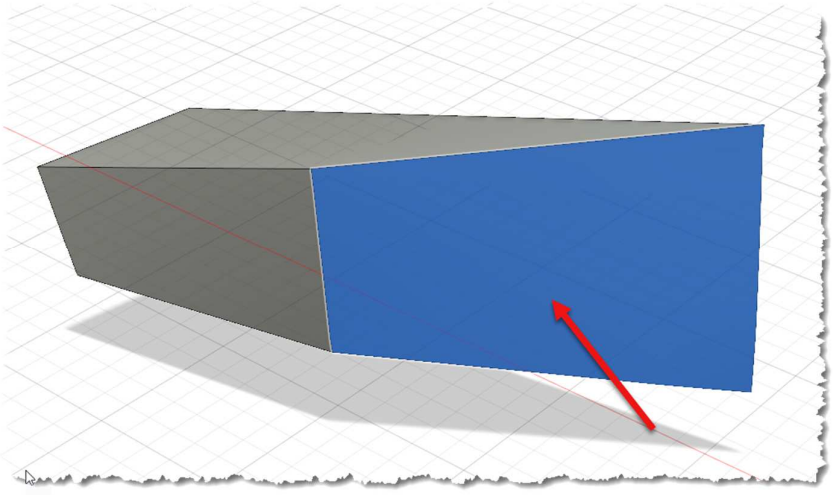
3. A single body is now shown on the project browser; this is the basic shape which will generate the external shell. The next step is making an offset outward of this *polysurface*;



4. Select the polysurface and specify an offset of 700 mm outward; click “Ok” when done;

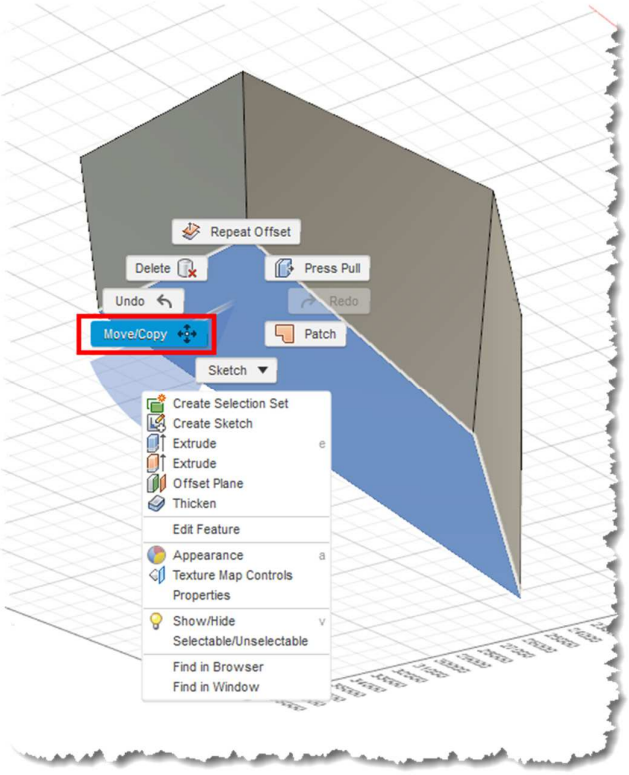


5. Remove the back face by selecting it and pressing the “Delete” key;

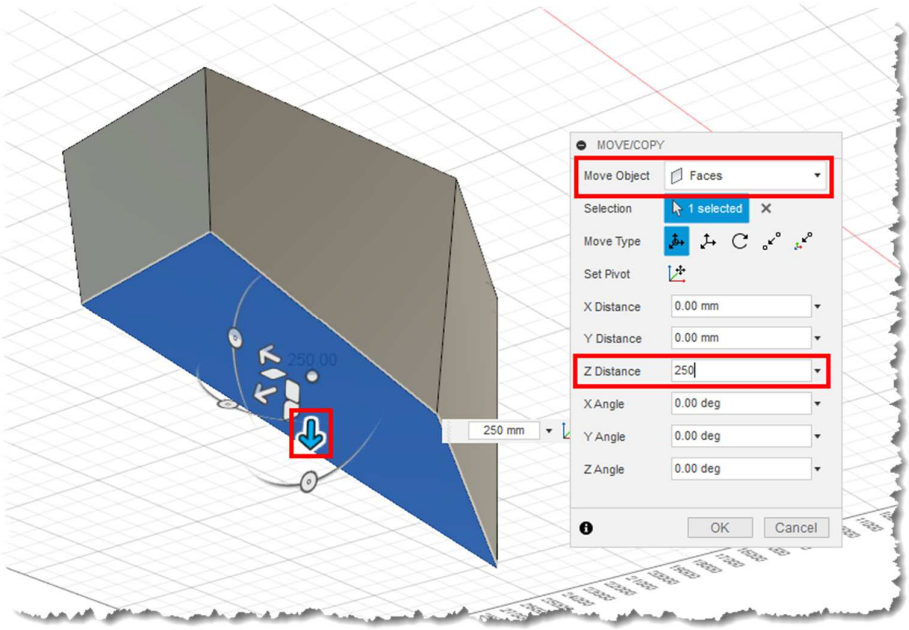


6. Switch off the visibility of the outer shell and repeat the offset command by 250 mm inward;
7. Leave visible the inner shell only;

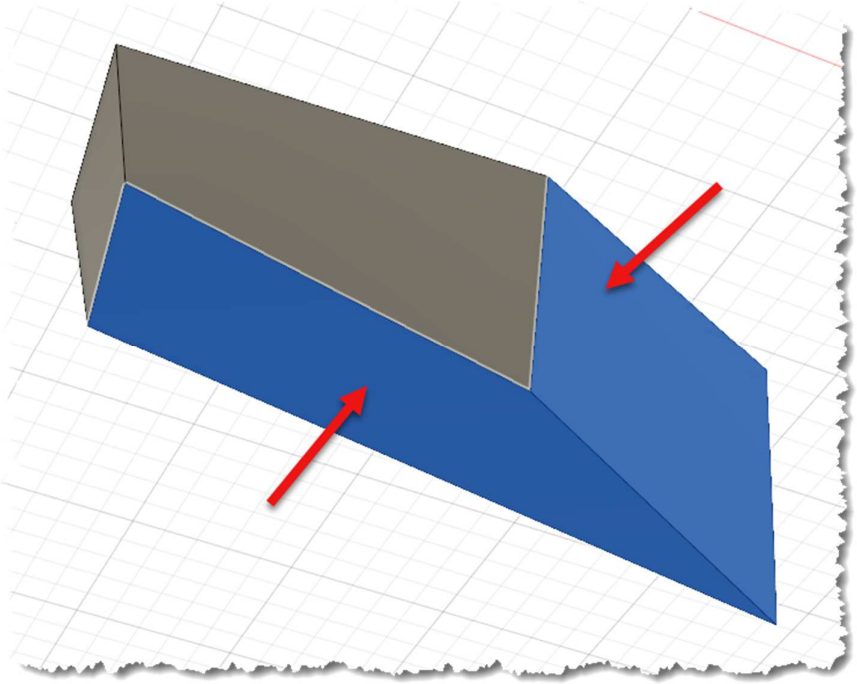
8. Select the bottom face and right click; select the “Move/Copy” command from the contextual menu;



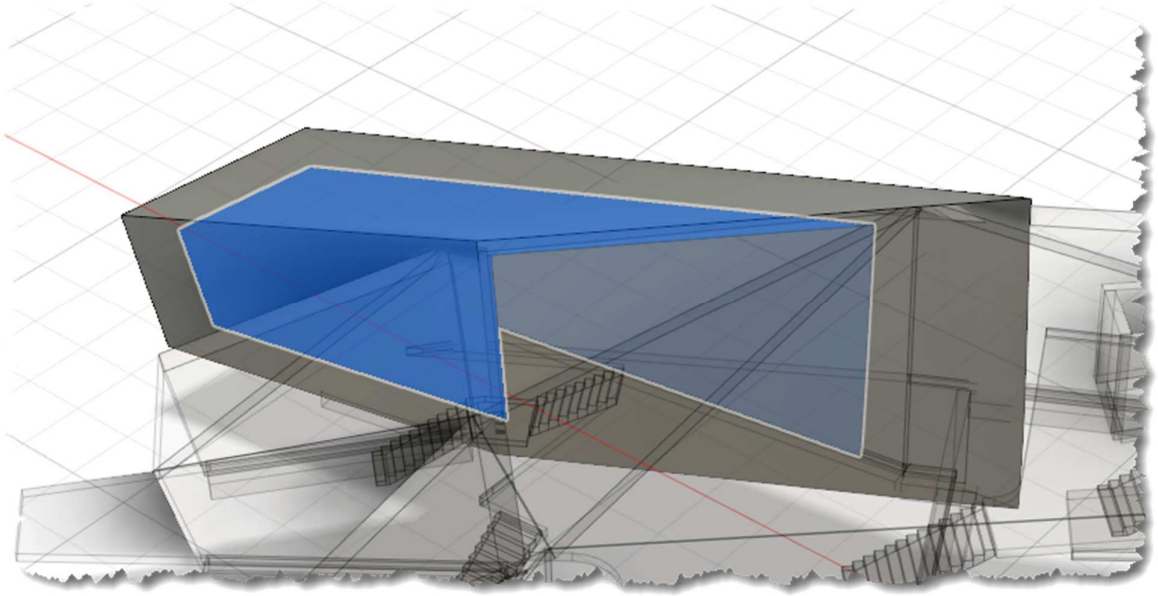
9. Specify that the object to move is a “Face”, then click on the bottom face to place the gizmo and specify 250 mm as Z movement; this command brings back the bottom face at the original position in order to have it coplanar with the building flor;



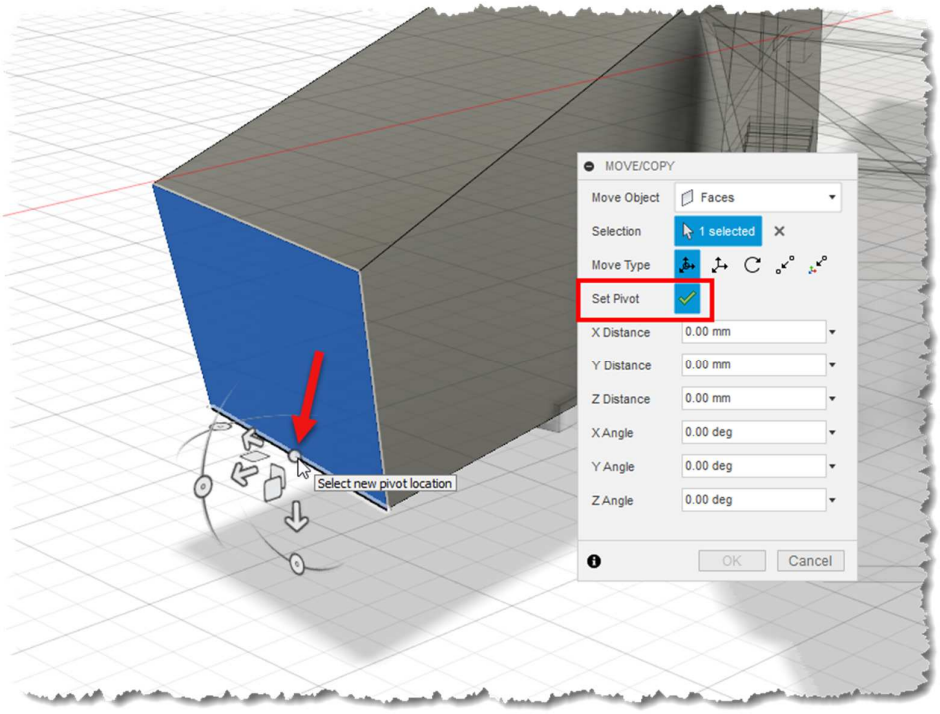
10. Remove the bottom and the back faces since they are not used to define the interior cladding;



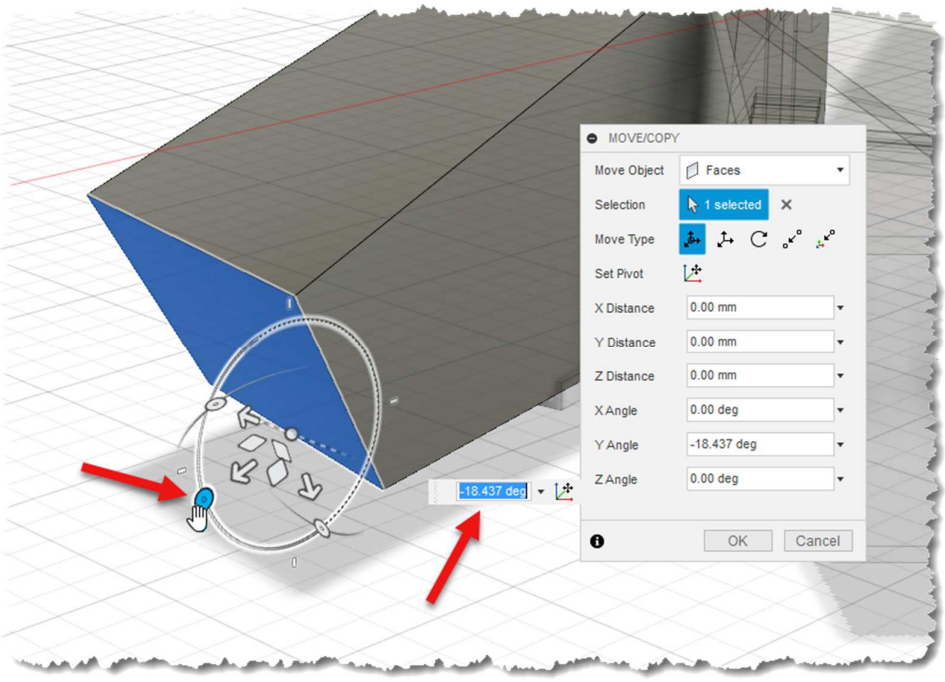
11. Following image shows the two shells;



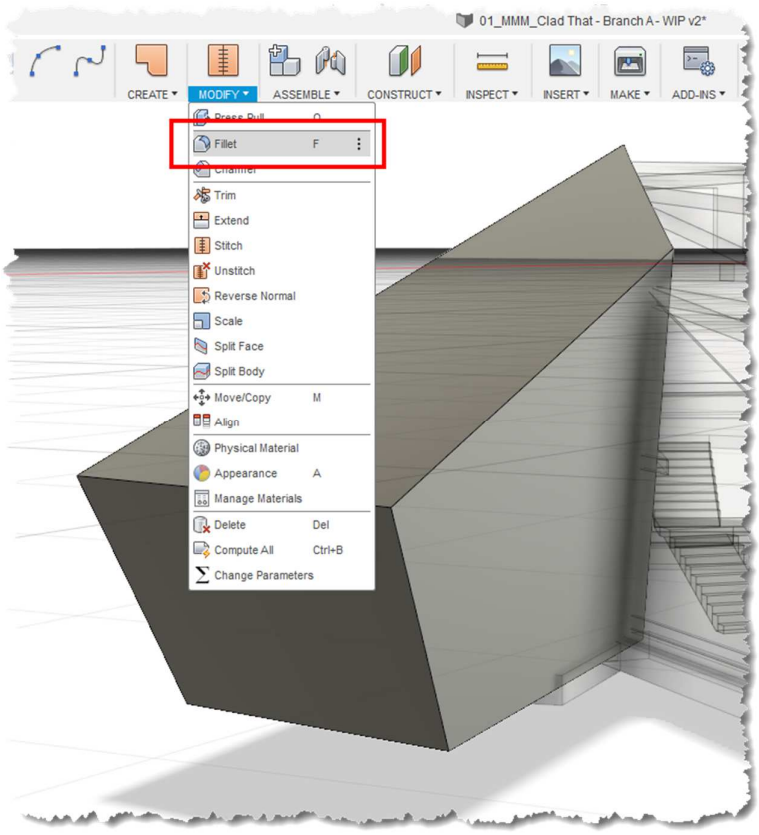
12. Use again the “Move/Copy” command to increase the slope of the front face; select the face and set the pivot on the bottom edge as shown on the picture below; click on the green checkmark when the pivot is placed;



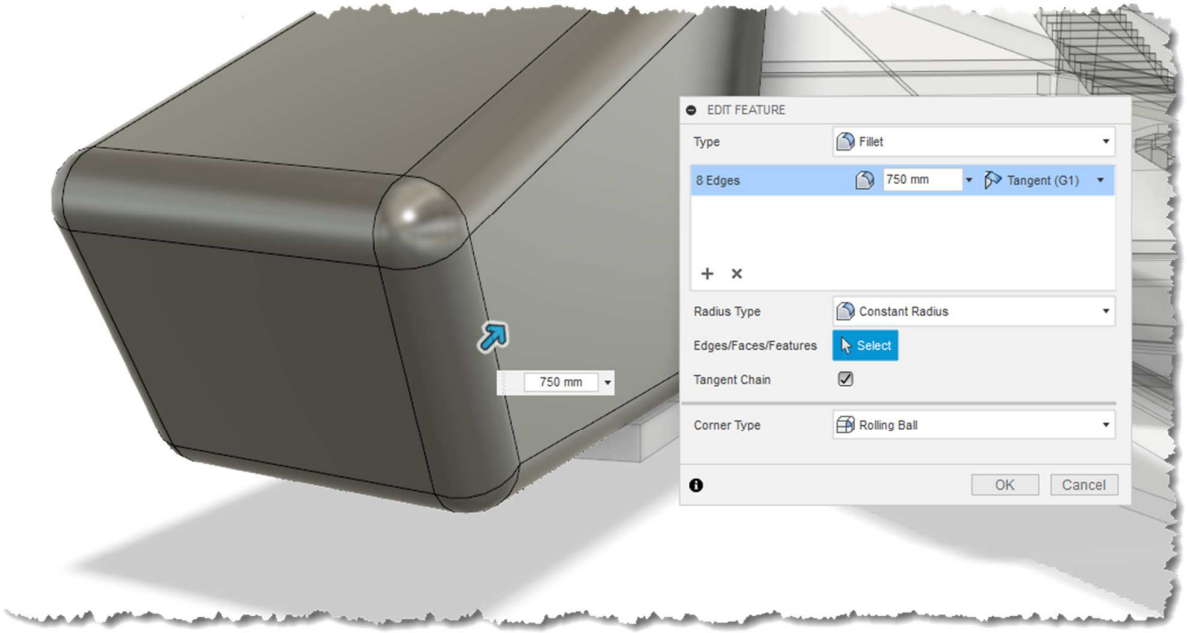
13. Select the rotation handle shown on the next image and rotate the face outward; it is also possible to specify an angle instead; click “Ok” when done;



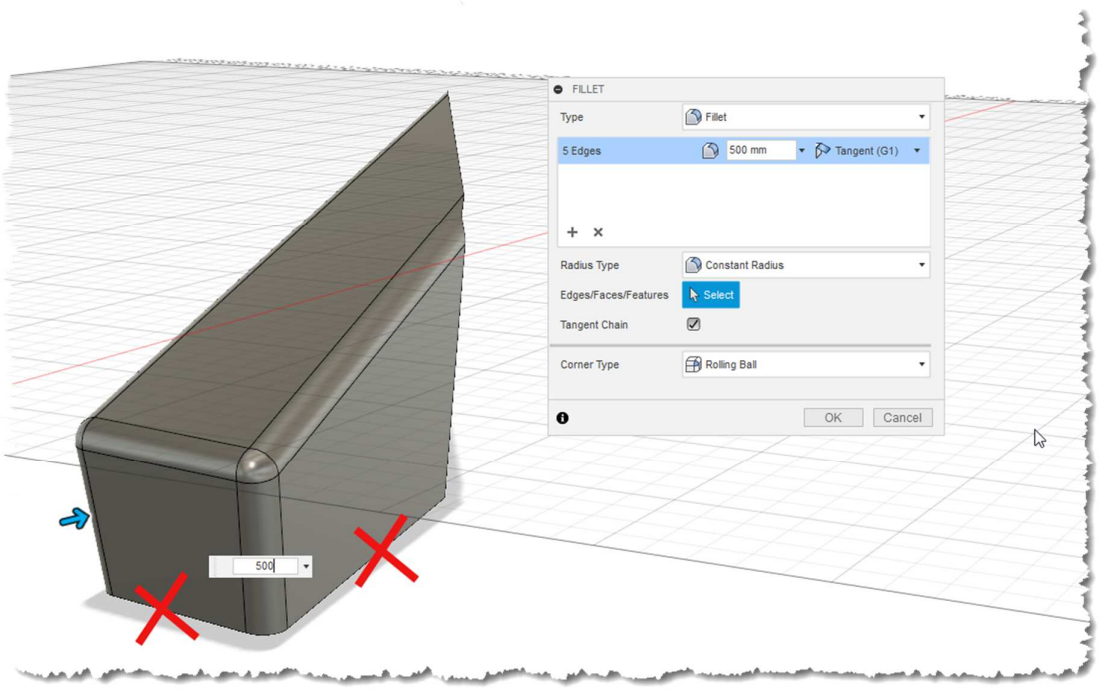
14. Use the “Fillet” command to create a round transition between the faces;



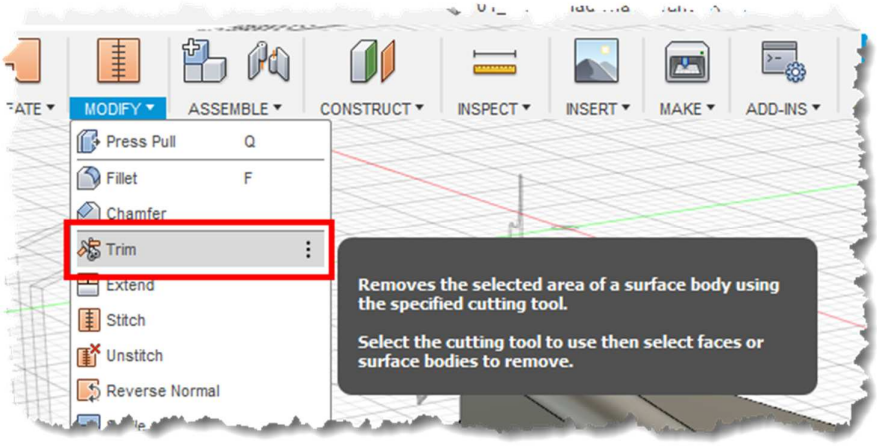
15. Specify a fillet radius of 750 mm;



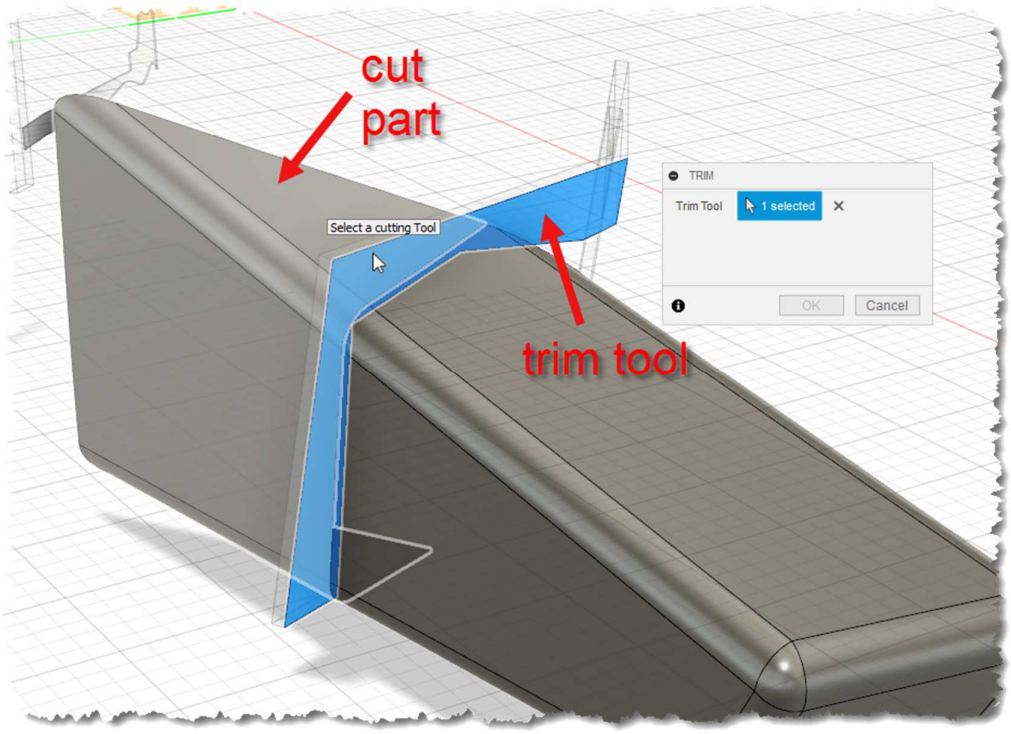
16. Turn off the outer shell and repeat the same on the inner geometry, specifying a fillet radius of 500 mm; here five edges only have to be filled being that two faces have been removed before;



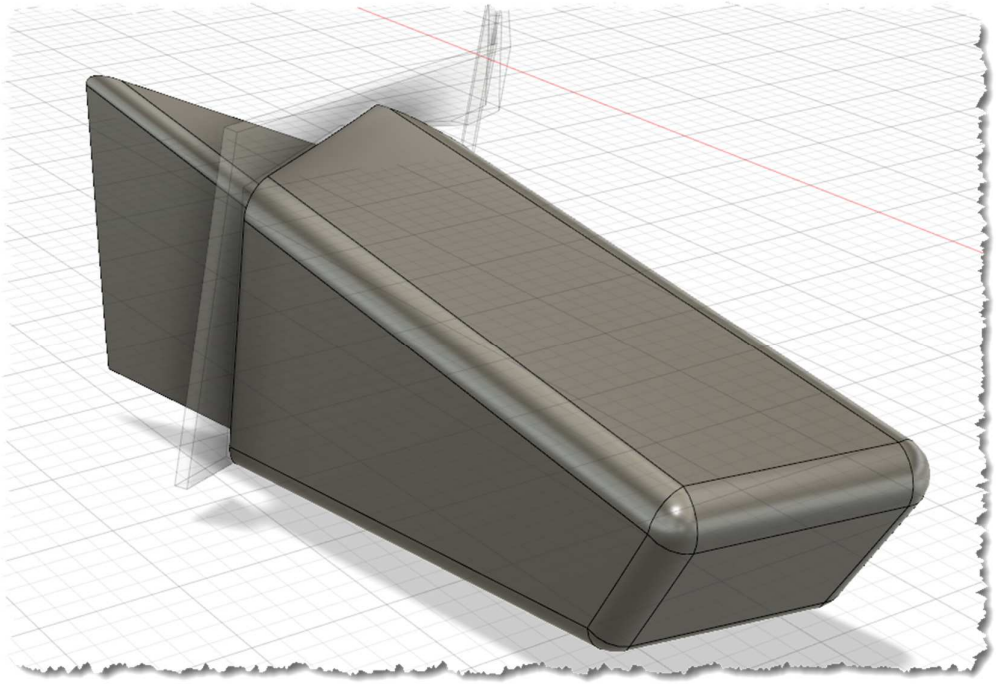
17. Turn on the outer shell and the component called “Boundaries”: this is related to retaining walls which limit the cladding extension;
18. Use the “Trim” command to remove the surfaces going beyond the retaining wall;



19. Use the Trim command to remove the part shown on the next picture;

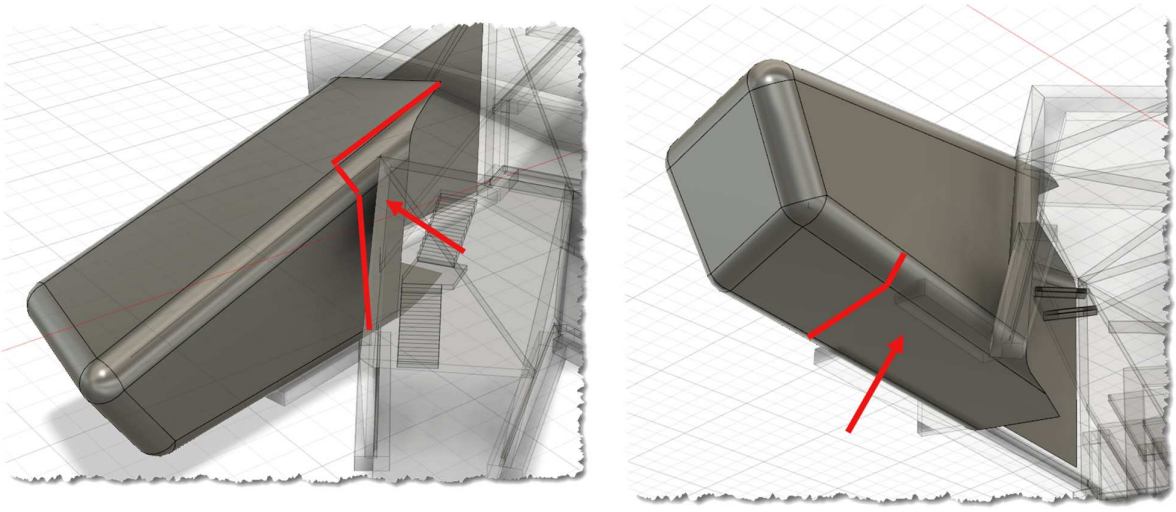


20. Click “Ok” when done; the result should be like that one on the following image;

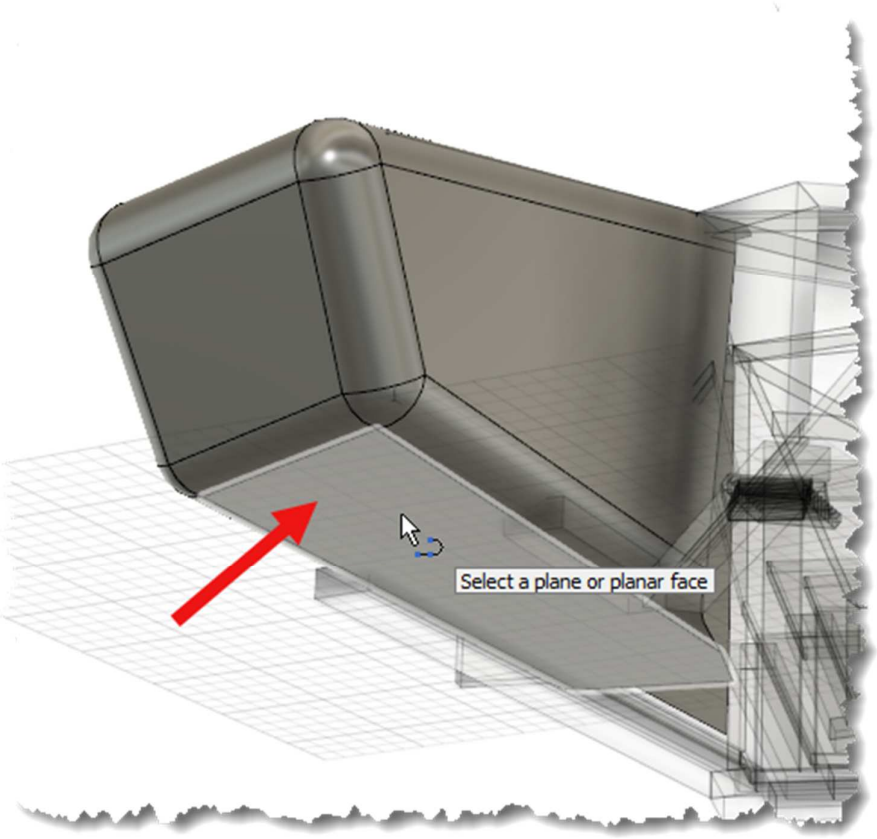


21. Before continuing with the cladding definition, a couple of more cuts have to be performed;

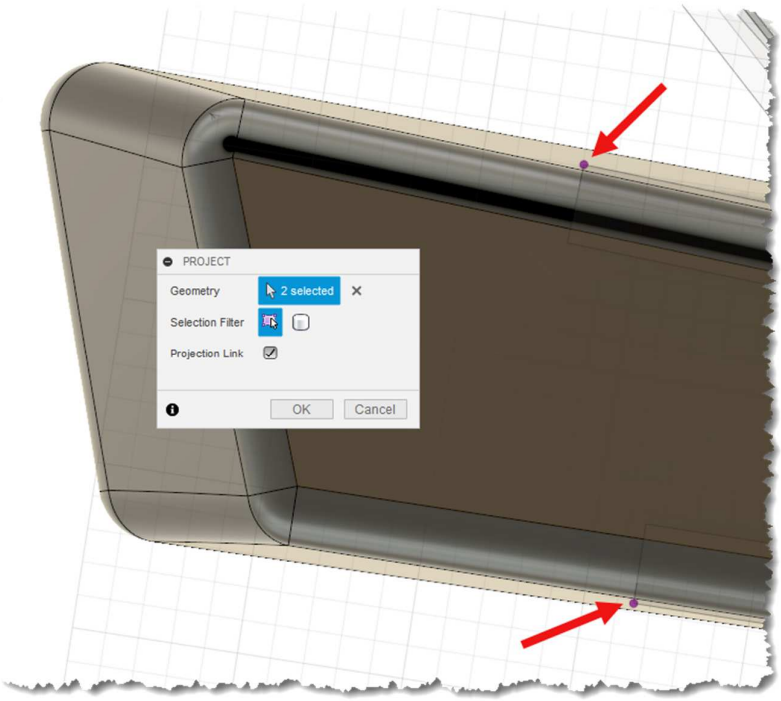
22. Turn off the visibility of the internal shell;



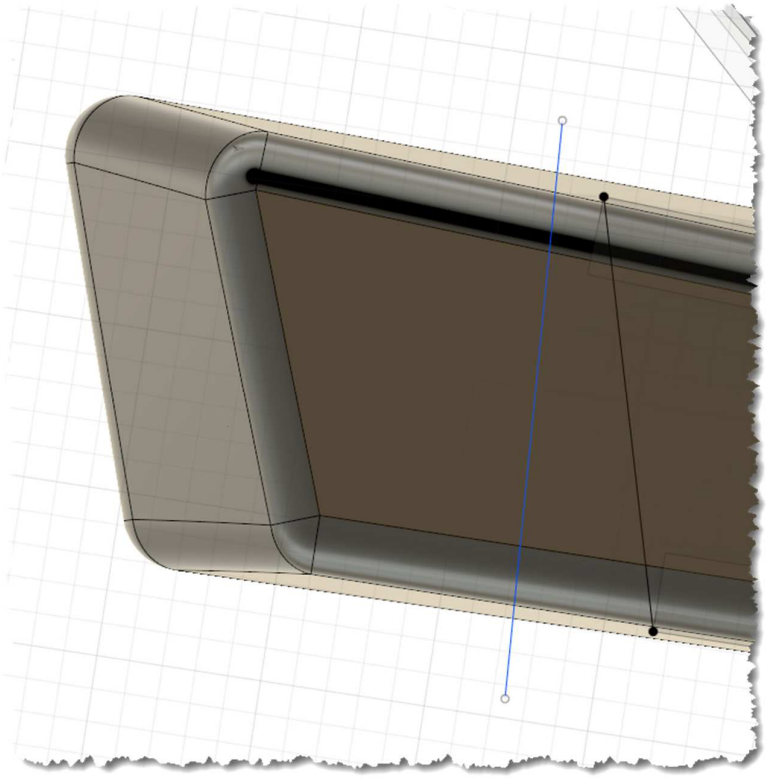
23. Create a new sketch using as a reference plane the bottom face from the outer shell;



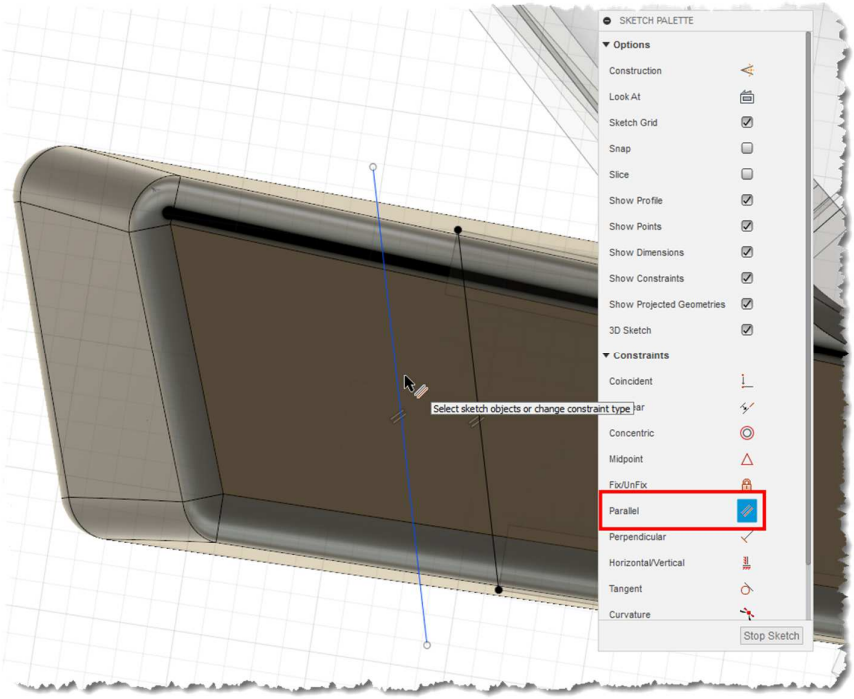
24. Use the “Project” command (under the “Sketch” menu) to include on the current sketch the two points of the continuous footing shown below;



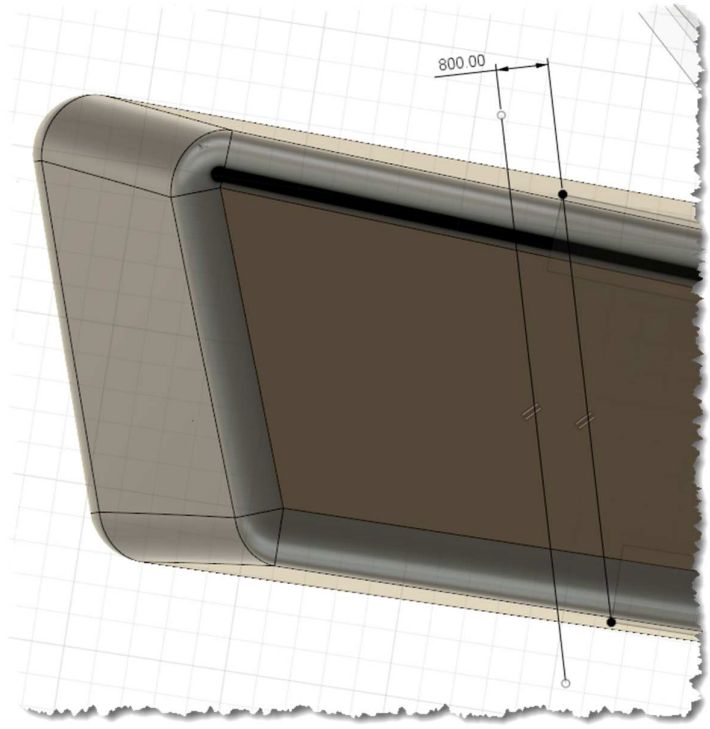
25. Create a line between these two points;
26. Create another line as shown below (the blue one);



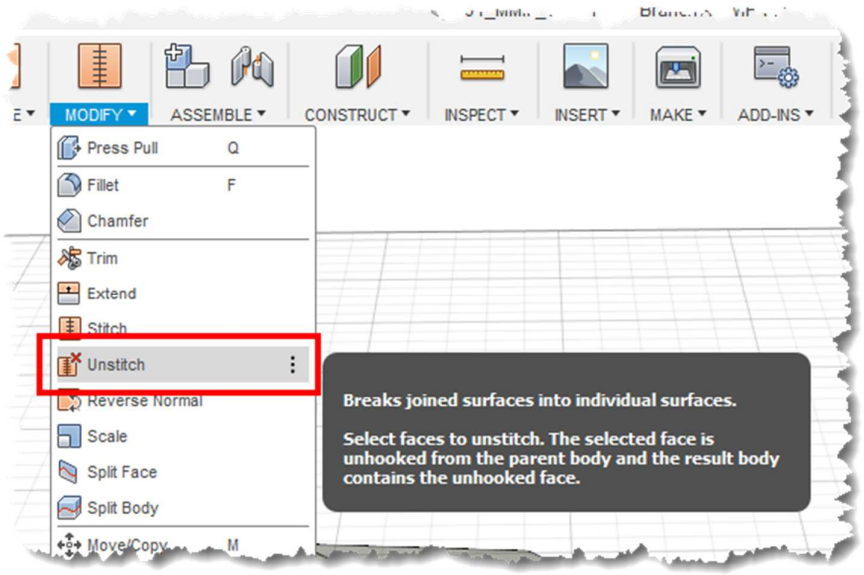
27. Use the “Parallel” constraint on the “Sketch Palette” to force the blue line to be parallel to the reference one. Select the reference line first then the second one to establish the parallel constraint;



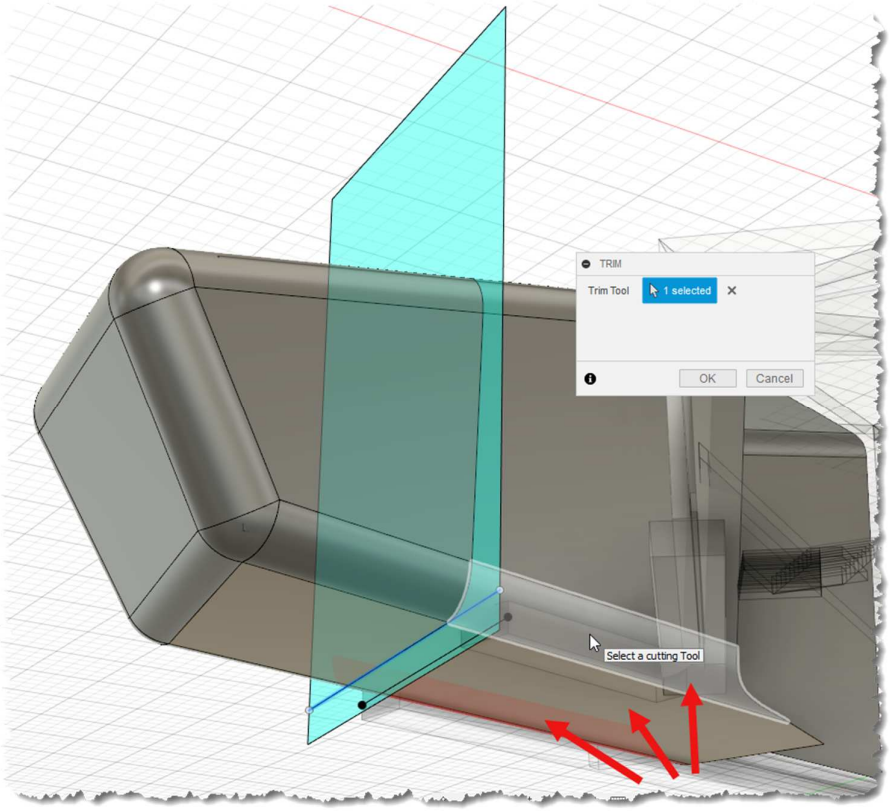
28. Create a dimension (use the “D” keyboard shortcut) between the two lines and specify 800 mm as the distance;



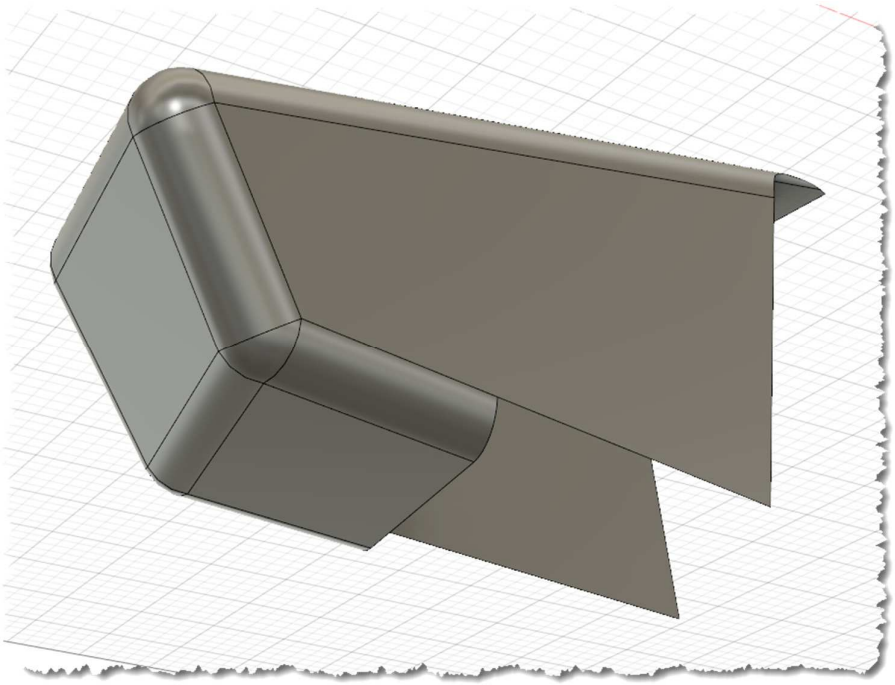
29. The blue line has become black since it is fully constrained now; click on “Stop Sketch”;
30. Unstitch the outer shell to divide it on its composing bodies;



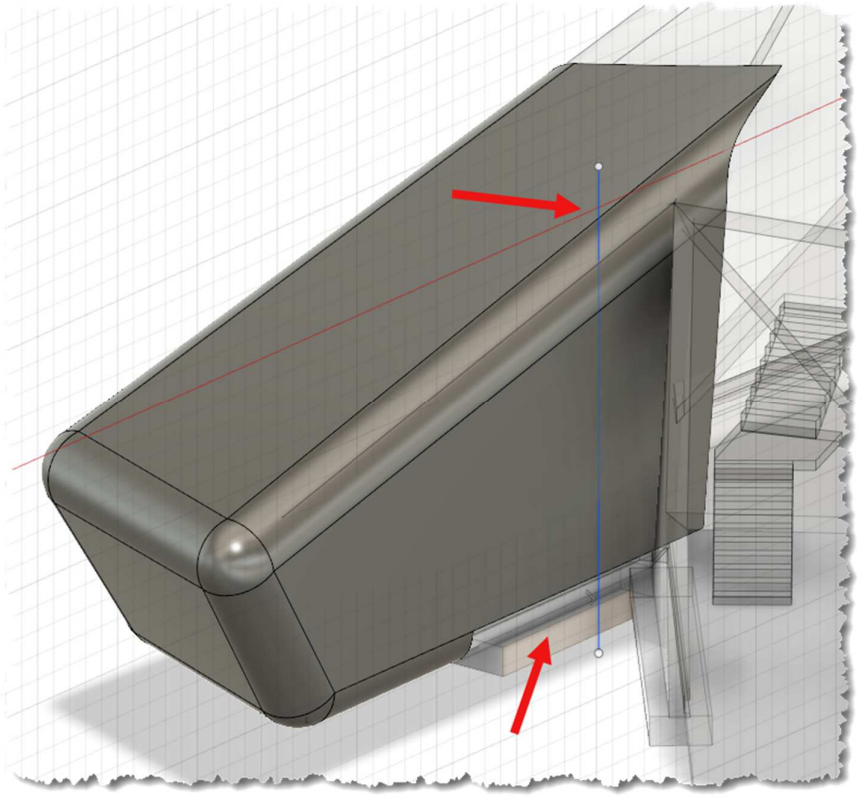
31. Use the “Trim” command and select the new line as trim tool; remove then the bottom face and the two fillets behind this line. Note that the line acts as a surface whose extrusion direction is perpendicular to the sketch plan;



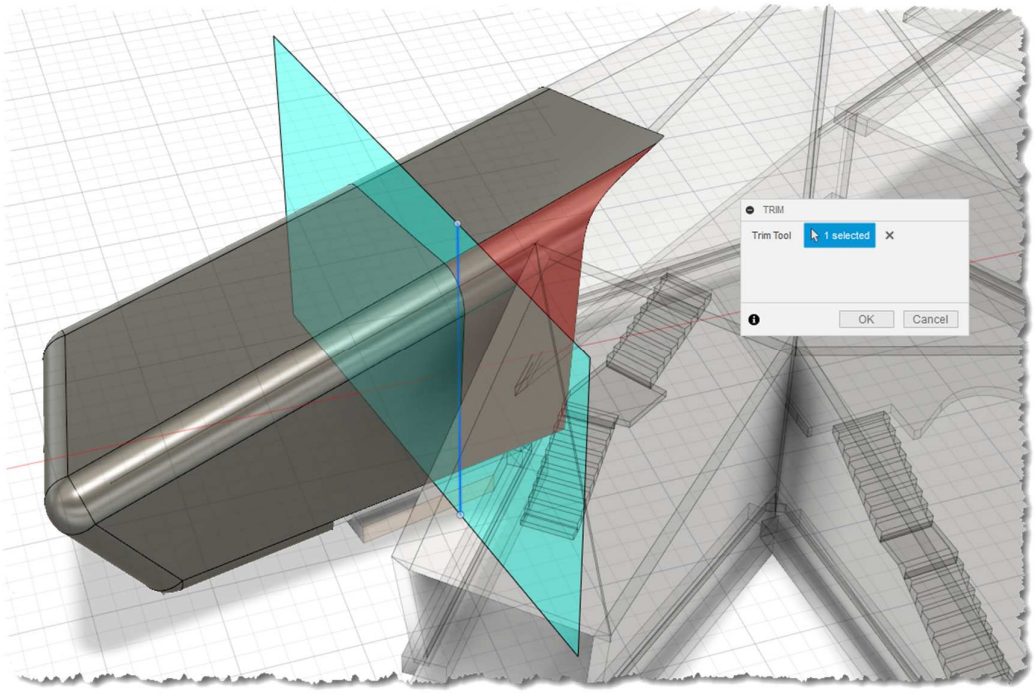
32. The cut bodies will appear as shown below;



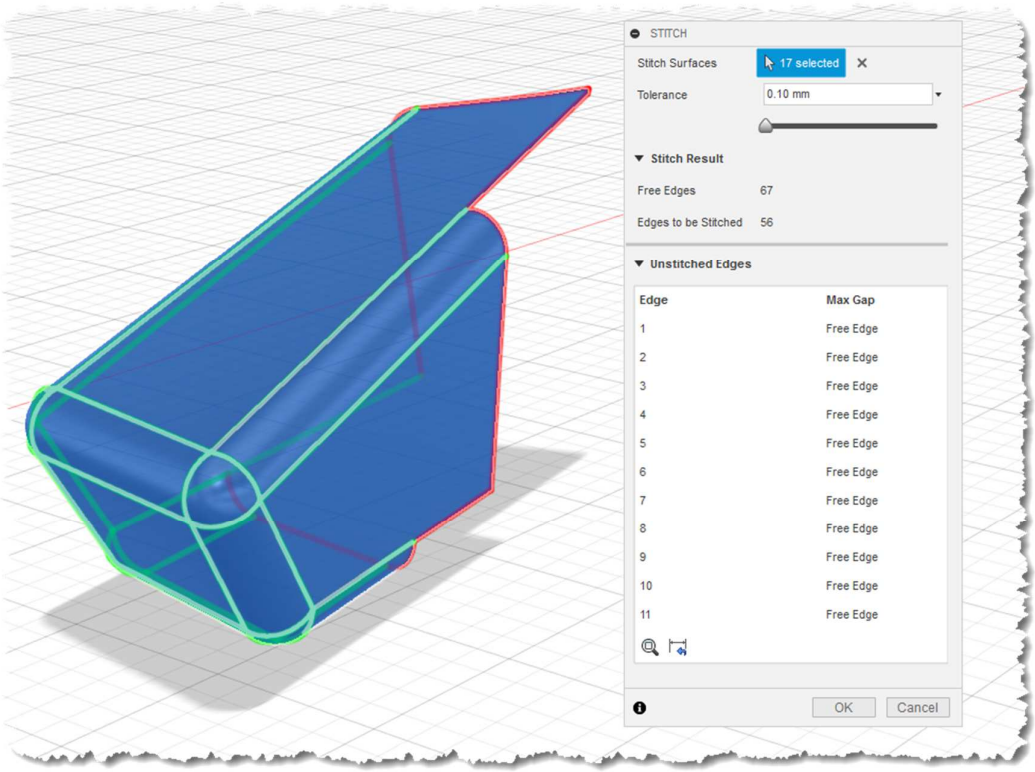
33. Repeat the same process using the indicated footing face and sketching a vertical line this time;



34. Use the line as the trimming tool to remove the portion of vertical face and its related fillet;

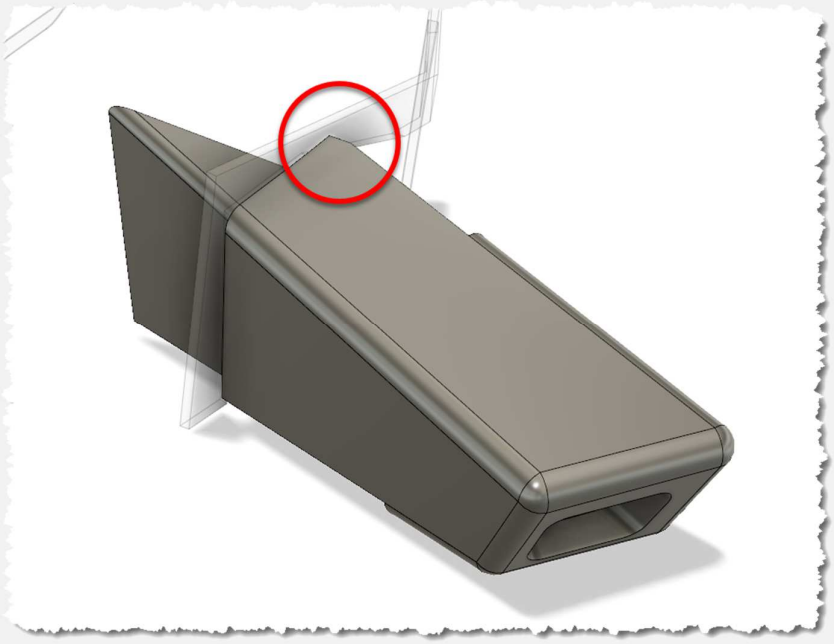


35. Stitch again the outer bodies to recreate the single shell;

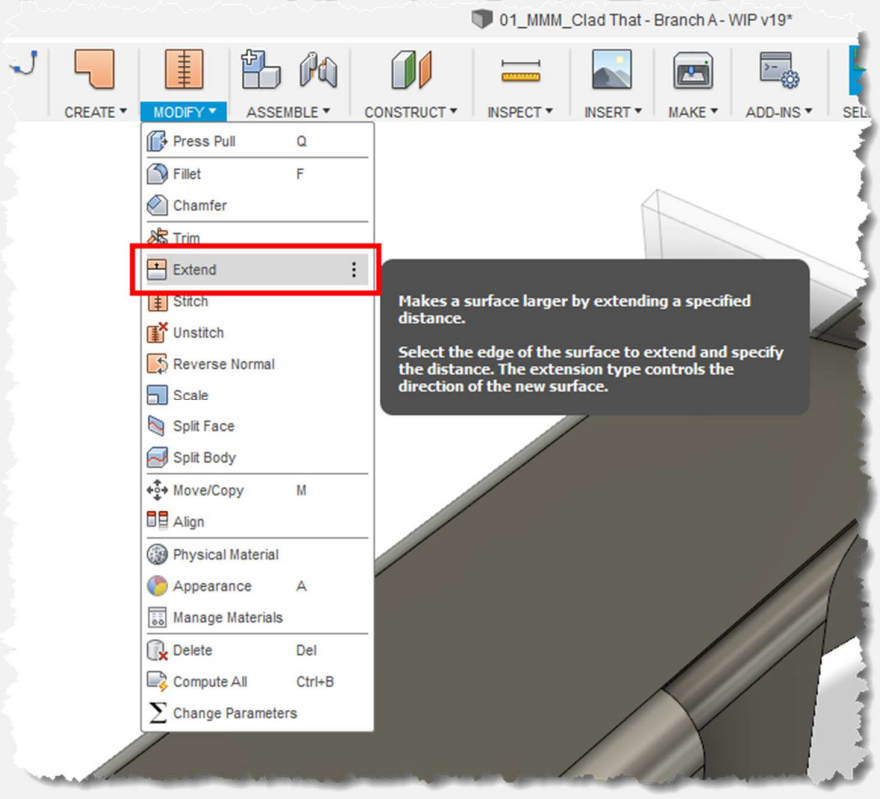


OPTIONAL

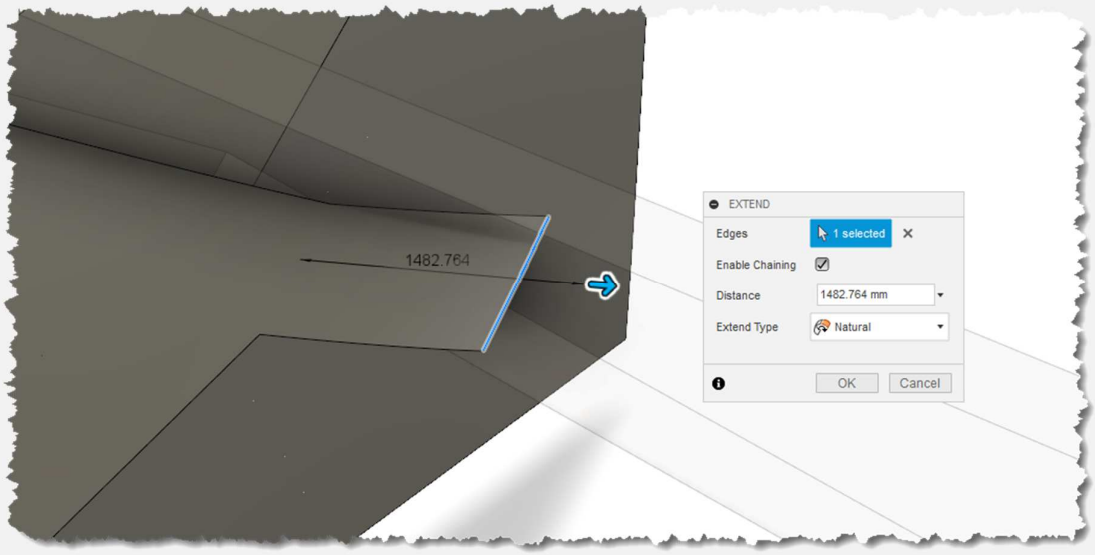
It can happen the case that the created surfaces need to be “fine-tuned” or refined. In this current case, the top face shape could be improved by adding a triangular part to better reach the retaining wall.



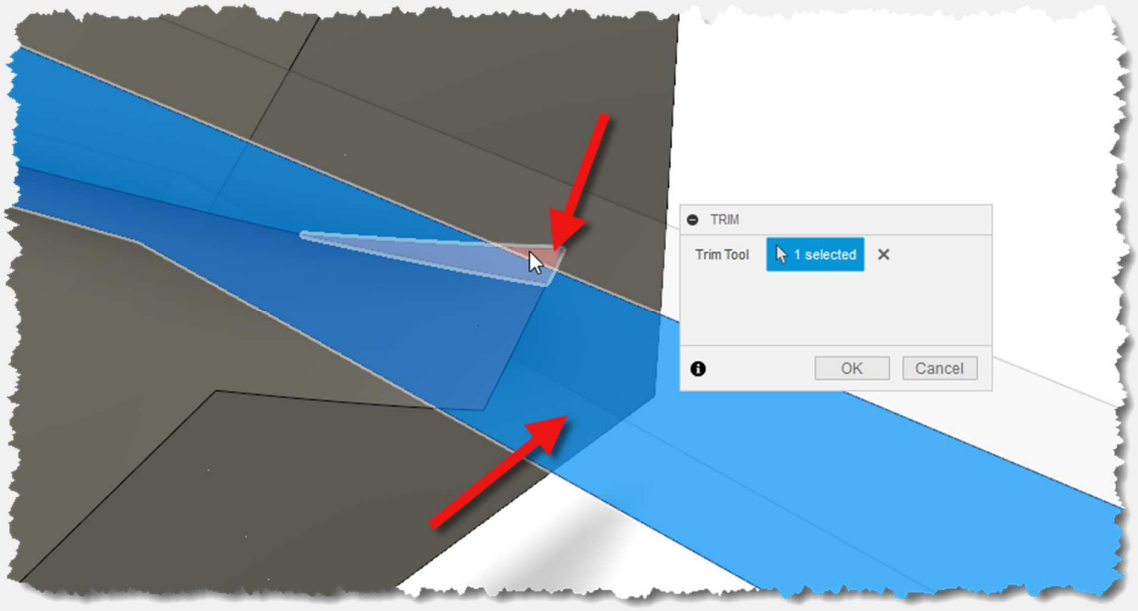
1. Use the “Extend” command under the “Modify” menu, “Patch” workspace;



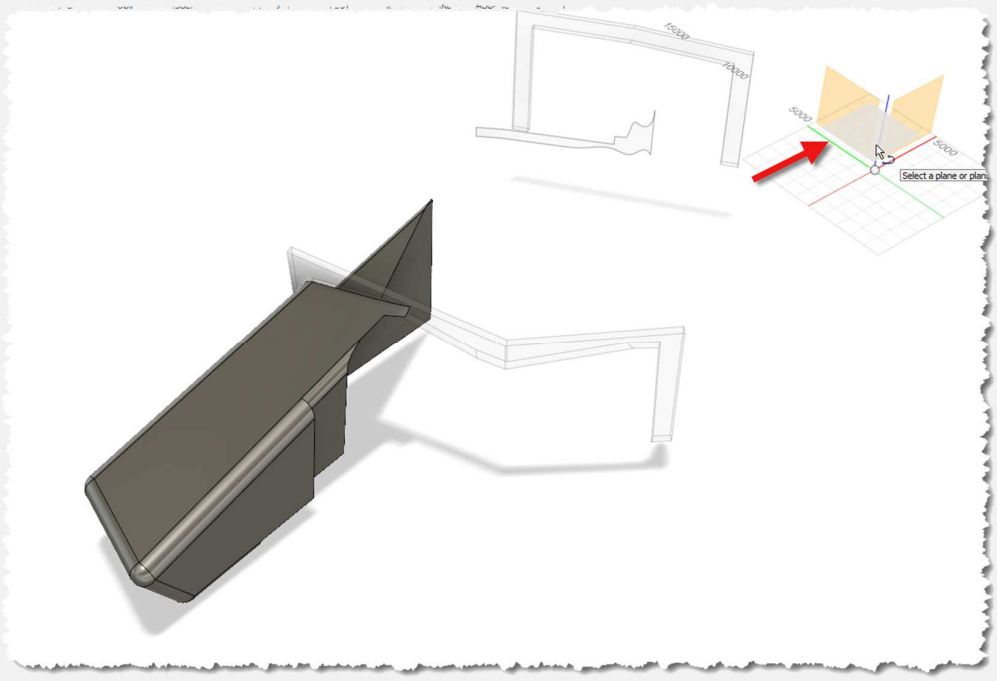
2. Select the highlighted edge and extend it so that it intersects the retaining wall;



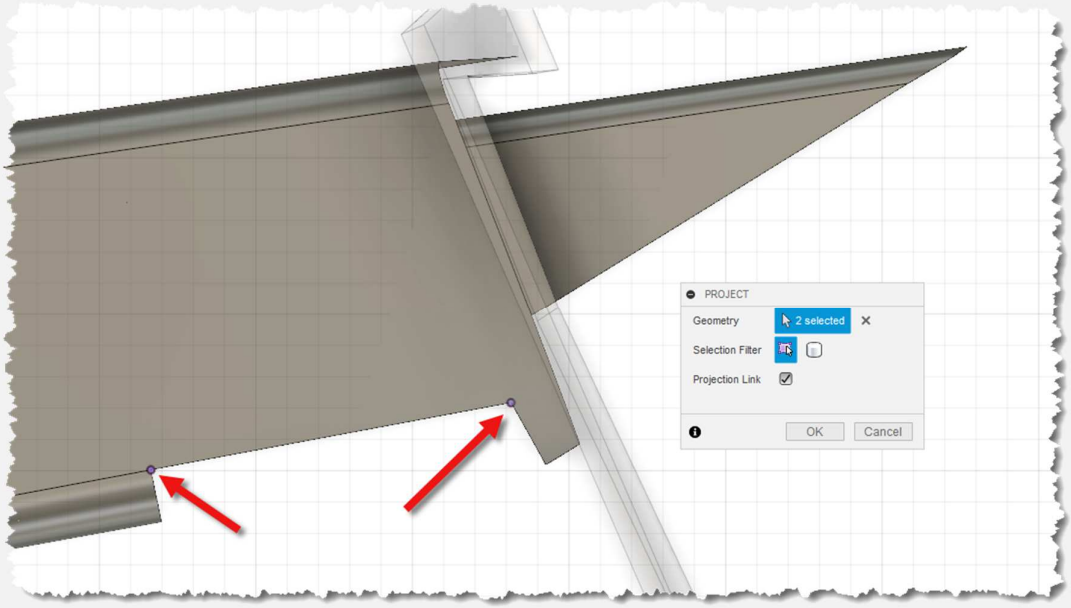
3. Use the “Trim” command to cut the face portion beside the retaining wall;



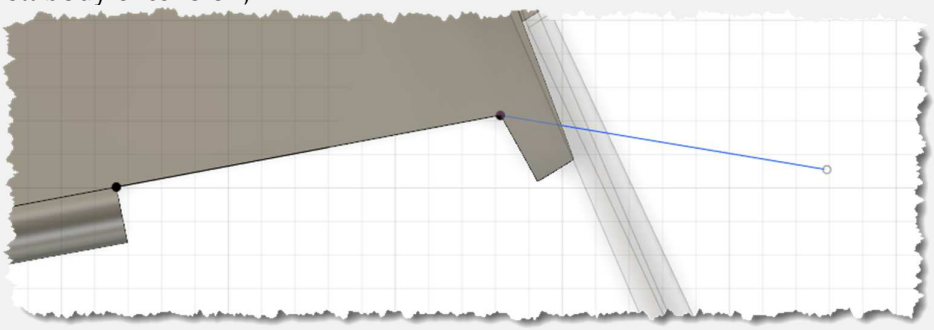
4. Create a new sketch using the XY place as the host;



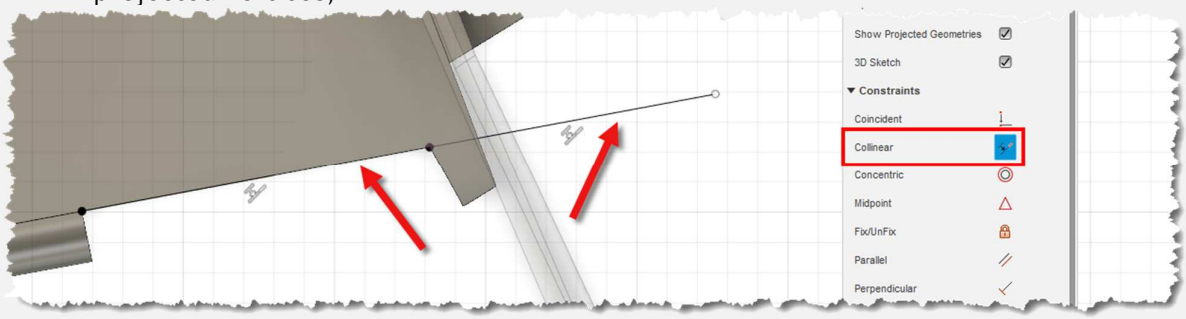
5. Use the “Project” feature to capture the two points highlighted below;



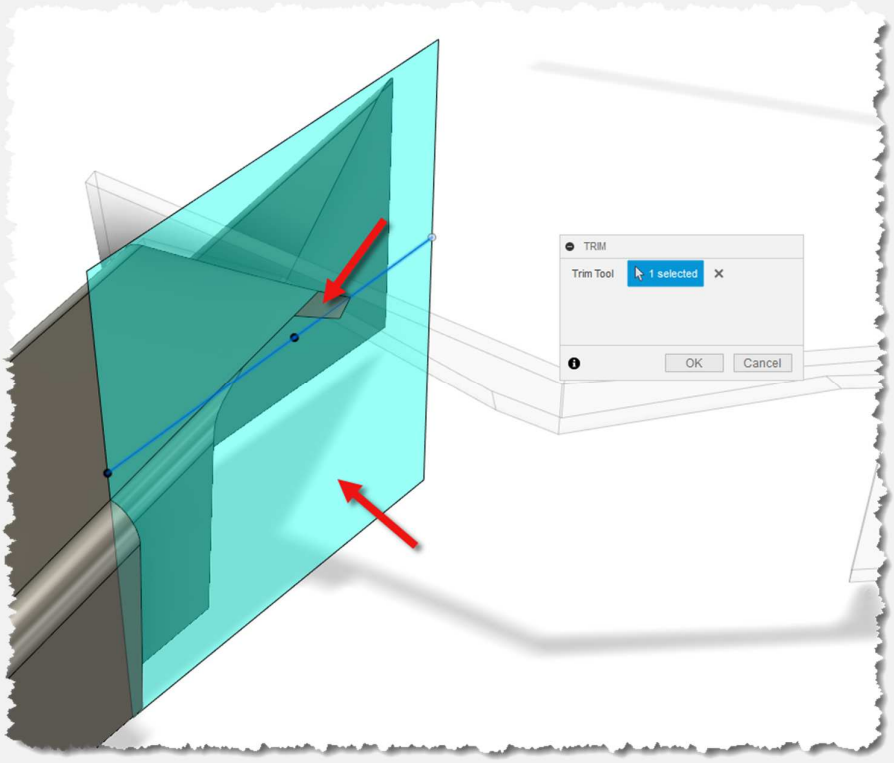
6. Create a line between these two points and an additional one which runs beside the shell body extension;



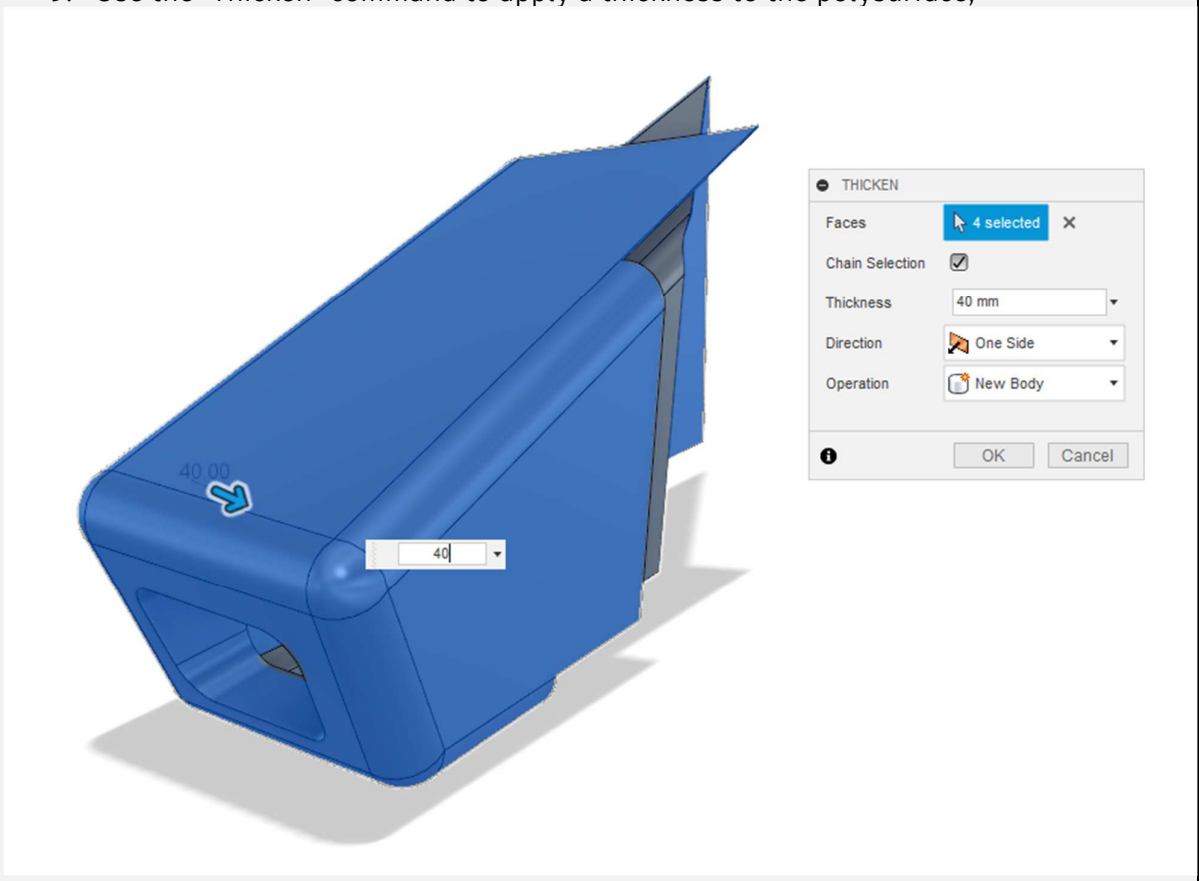
7. Use the “Collinear” constraint to align the additional line to that one between the projected vertices;



8. Use the line to cut the extended surface. Note that the line acts as it were a surface extruded with a direction perpendicular to the line host plane;

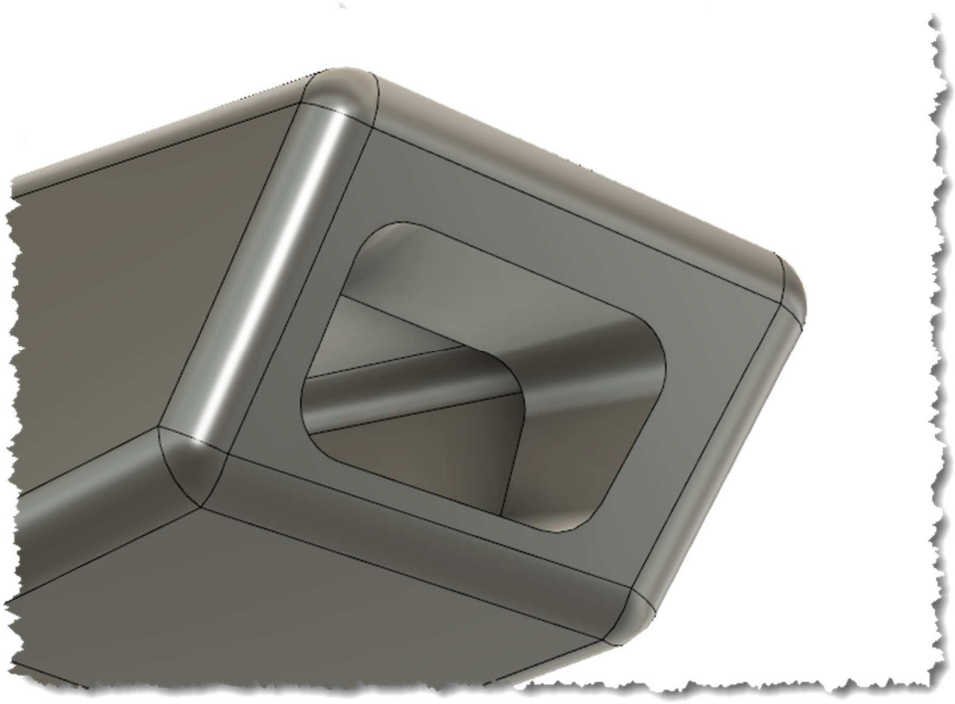


9. Use the “Thicken” command to apply a thickness to the polysurface;

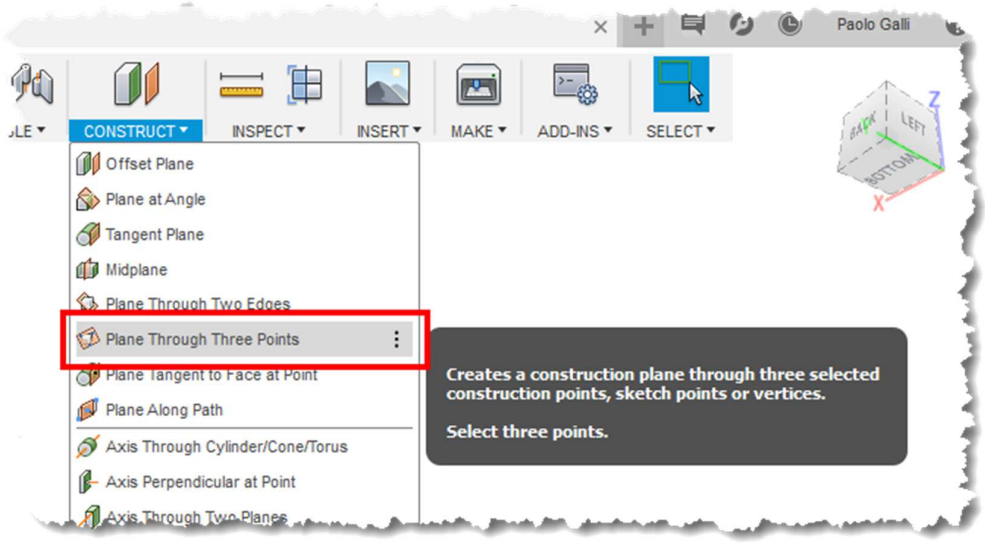


10.4 Window Opening Creation

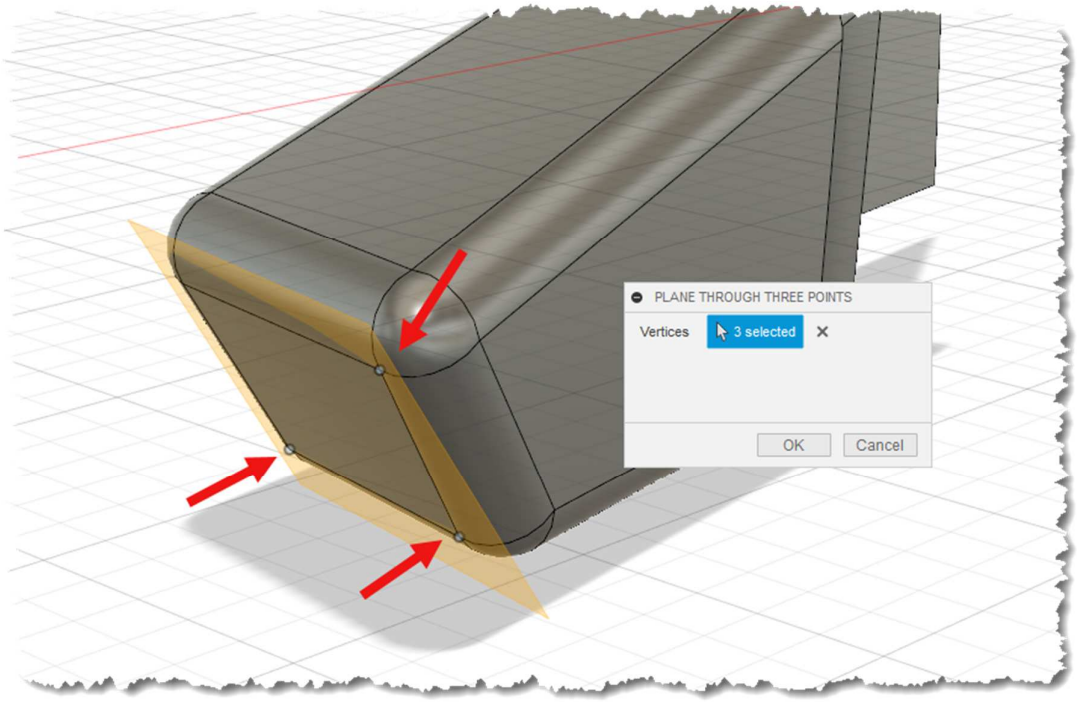
Once that both the internal and external shells have been modeled, the window opening must be defined. The desired shape, in this case, is a blend between a smaller internal profile and a bigger exterior one.



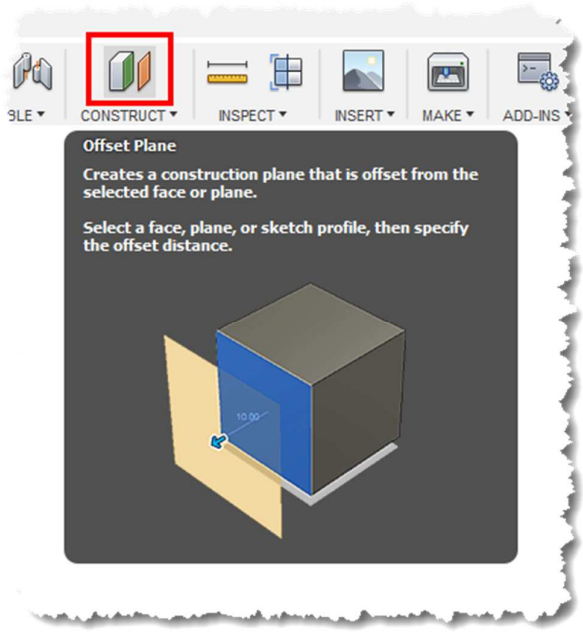
1. Turn on the visibility of both the internal and external shells;
2. Since the exterior face is not planar, a plane approximating its shape must be created in order to define a sketch on it. Being that three points can be identified, select the "Plane Through Three Points" command under the "Construct" menu;



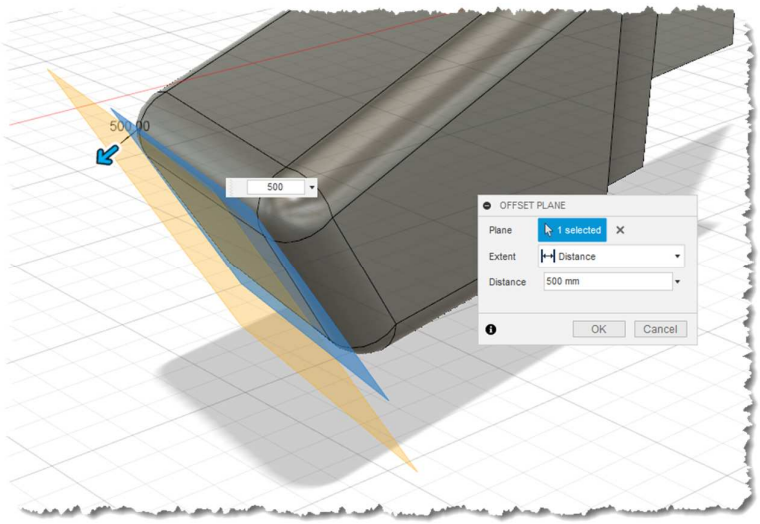
3. Select the points highlighted below and click “Ok” when done;



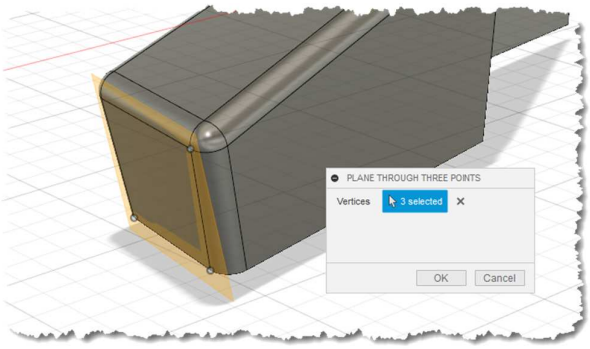
4. Use the “Offset Plane” command to create a parallel plane;

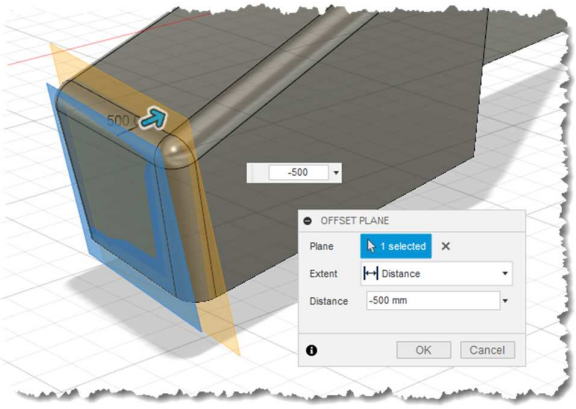


5. Specify 500 mm as the offset distance;

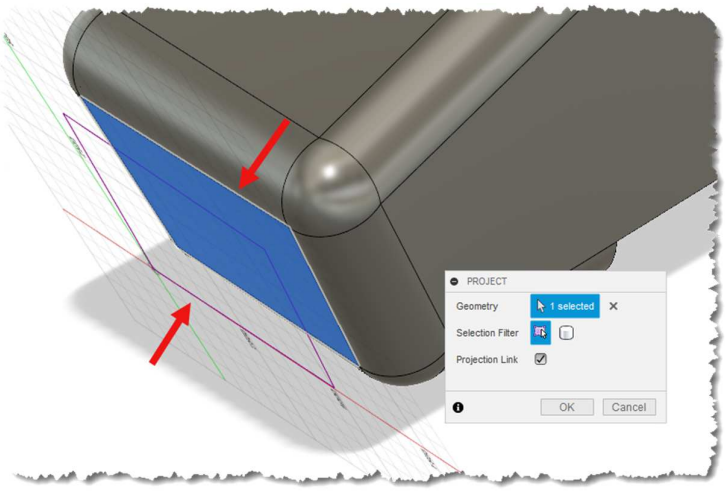


6. Repeat the same procedure for the internal shell ("Plane Trough Three Points" and "Offset Plane");

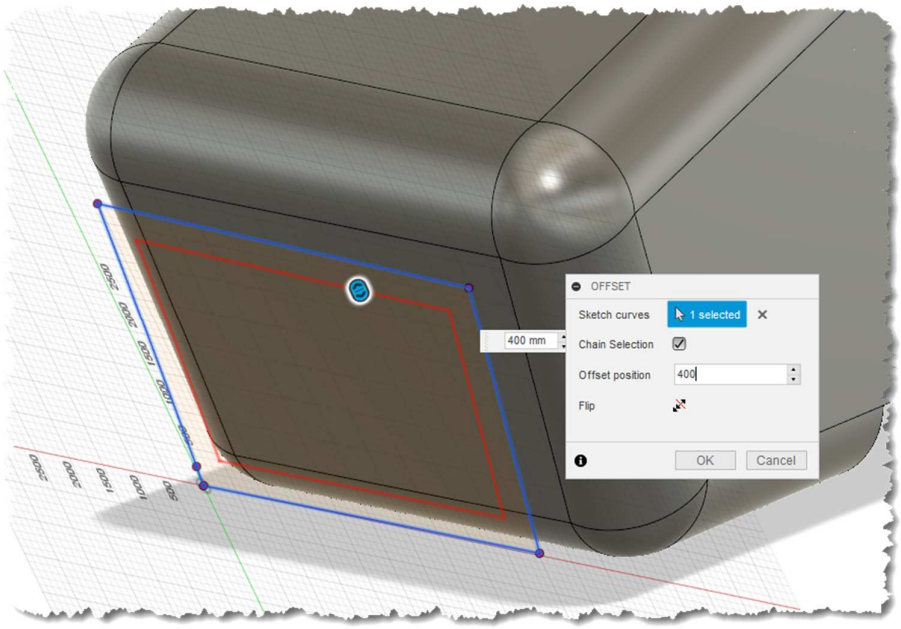




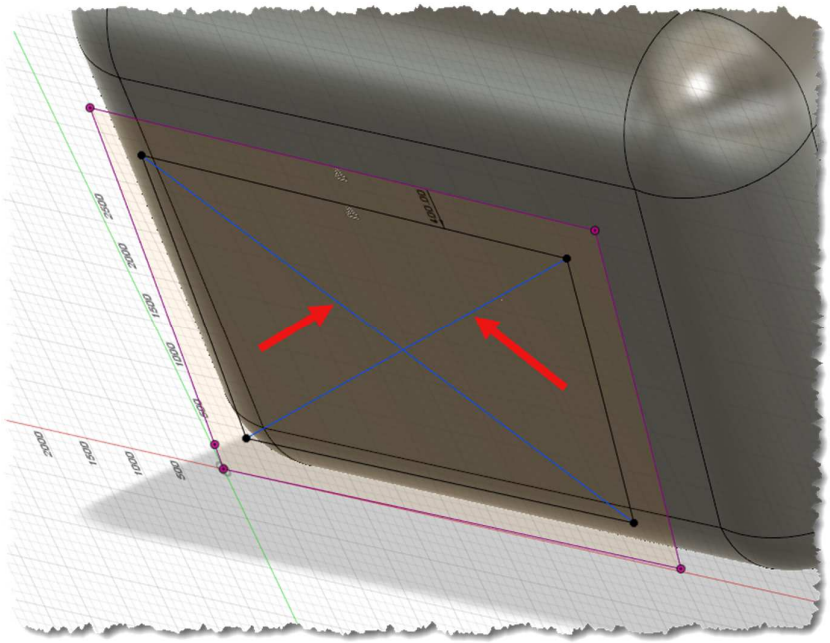
7. Create a new sketch using the outer plane and use the "Project" command to replicate the surface boundaries; select the whole face and click "Ok" when done;



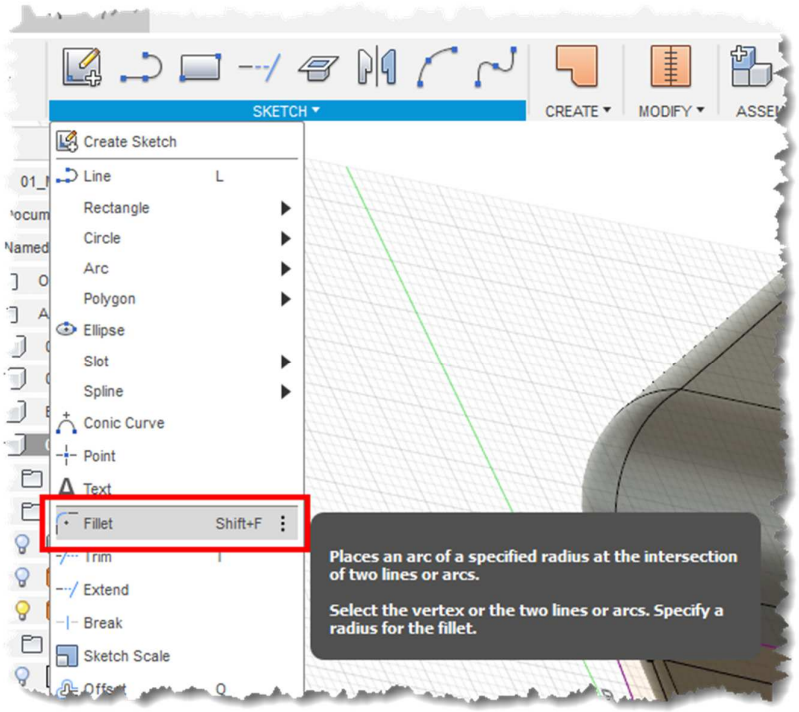
8. Create an internal offset with a distance of mm;



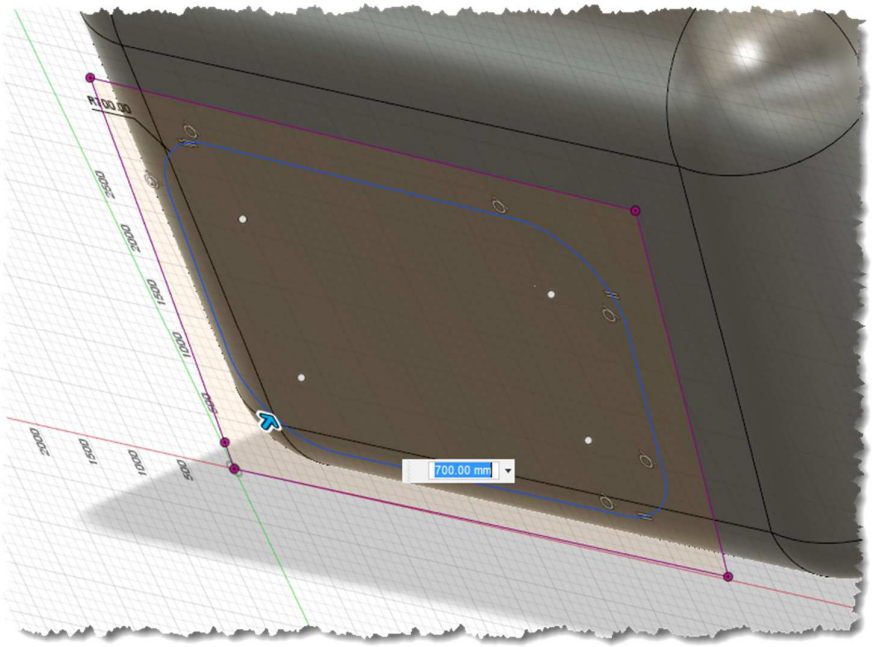
9. Being that the projected geometries could assume a shape which cannot be manipulated (for instance complex splines or similar curves), use the offset geometry to recreate a boundary composed of lines only. In this case a big “X” has been sketched as a reference, then the boundary lines have been deleted. The endpoints of the “X” have then been used to recreate 4 lines;



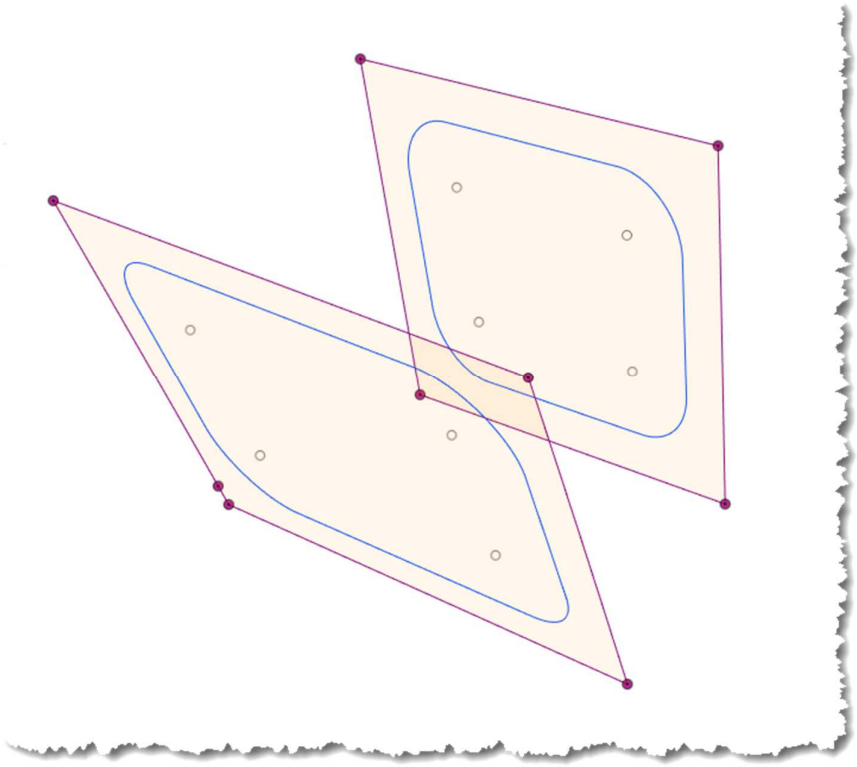
10. An alternative to this procedure is projecting the vertices only and then connect them with lines; the result of an offset of those lines would still represented by lines, so they can be easily manipulated;
11. Use the “Fillet” command to create a smooth transition between the edges;



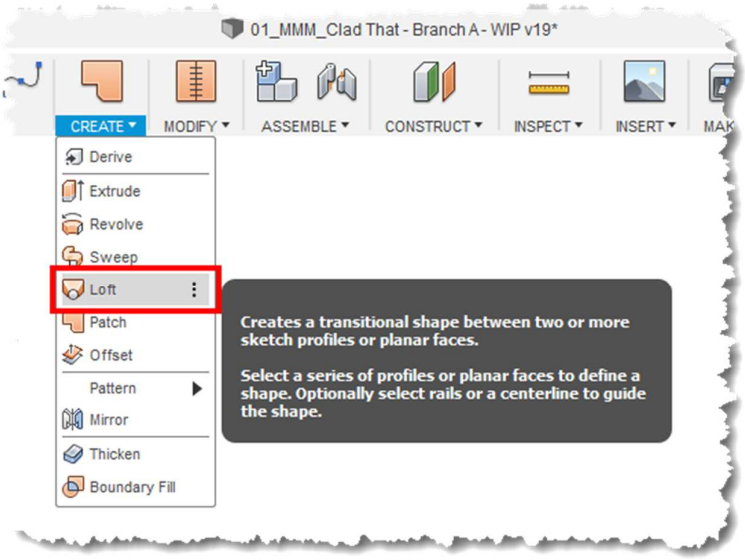
12. Specify a 700 mm fillet radius;



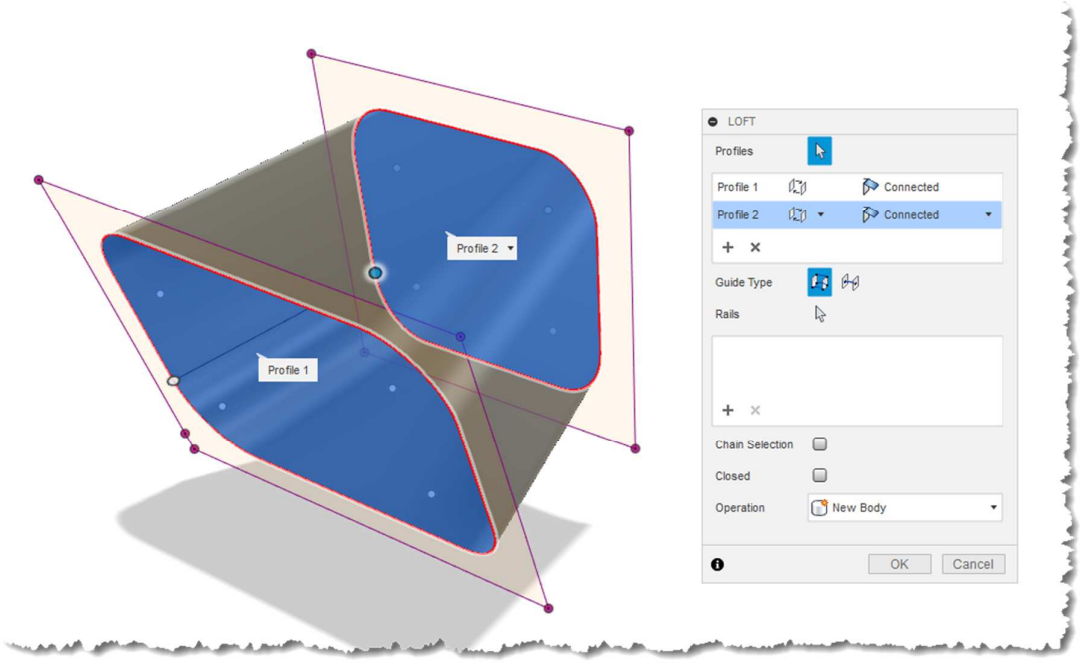
13. Repeat the same procedure for the inner plane; the final result should appear as in the next image;



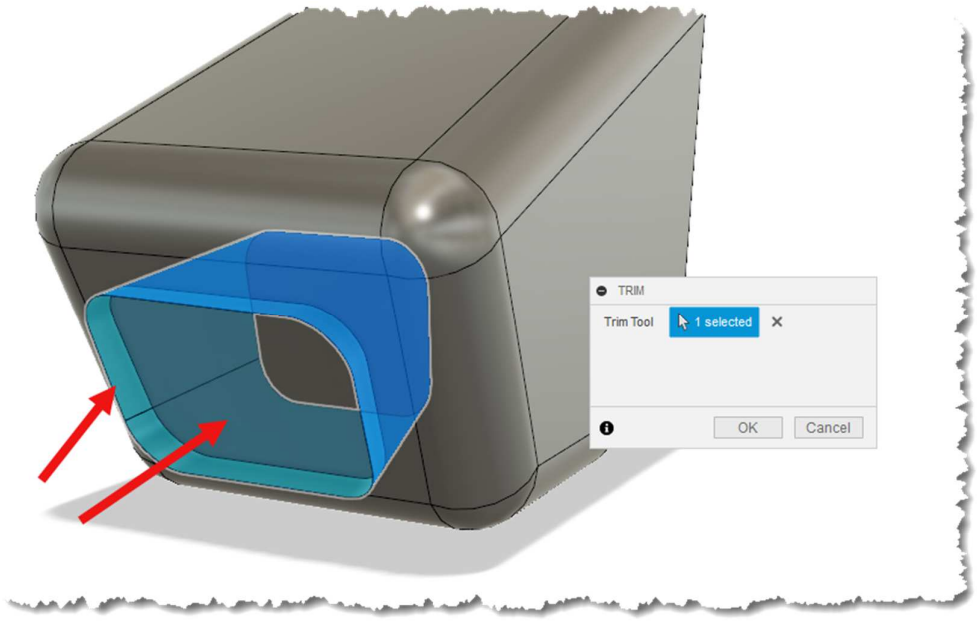
14. Use the “Loft” command under the “Create” menu of the “Patch” workspace;

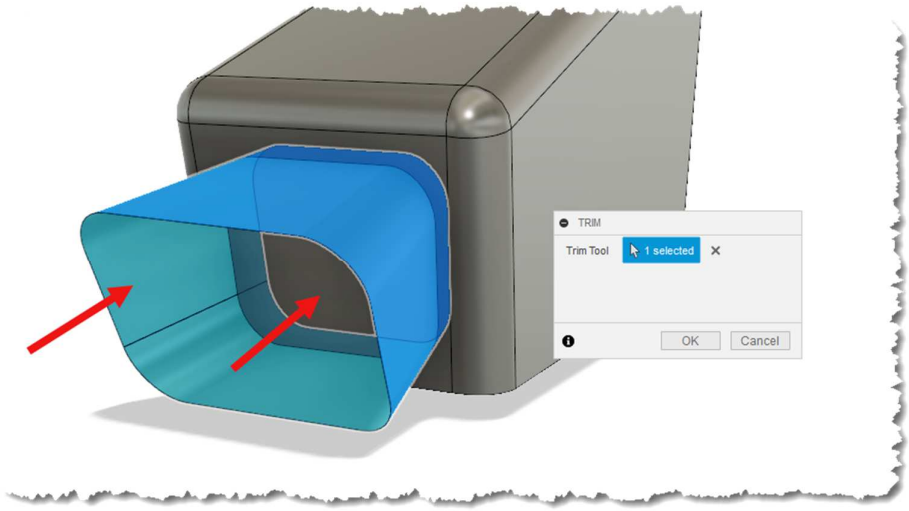


15. Select the interior and the exterior profiles to create a lofted shape; click “Ok” when done;

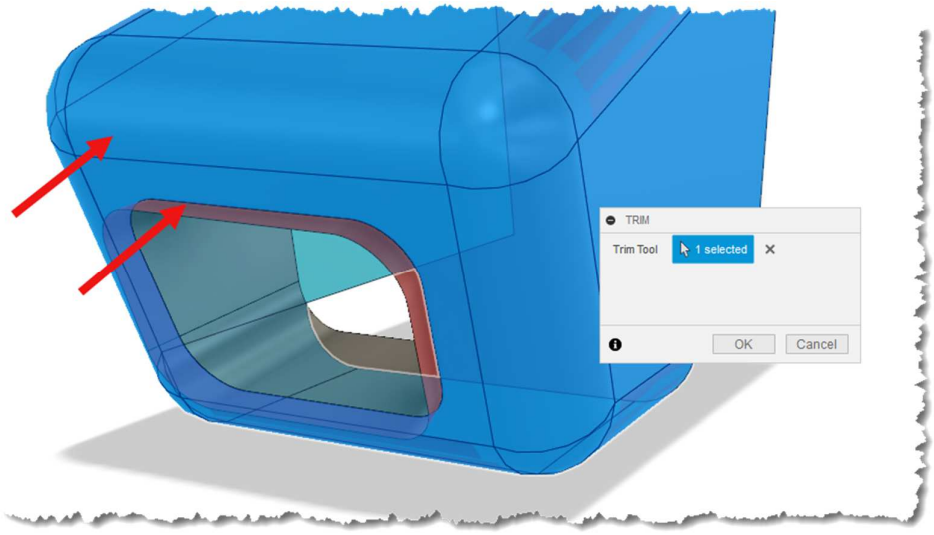


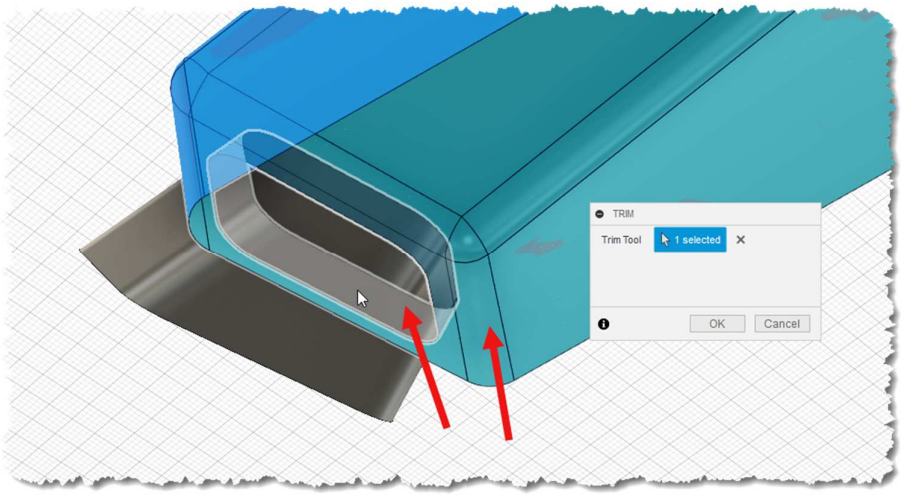
16. Use the “Trim” command to create the opening both in the external and in the internal faces;



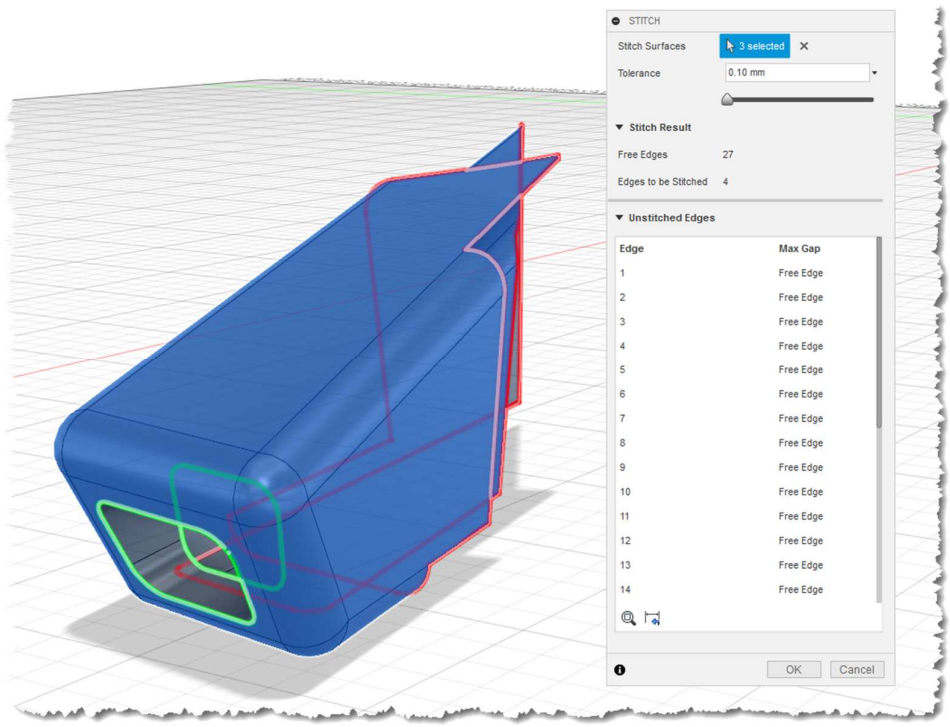


17. Repeat the same sequence to remove the loft geometry going outside the faces;

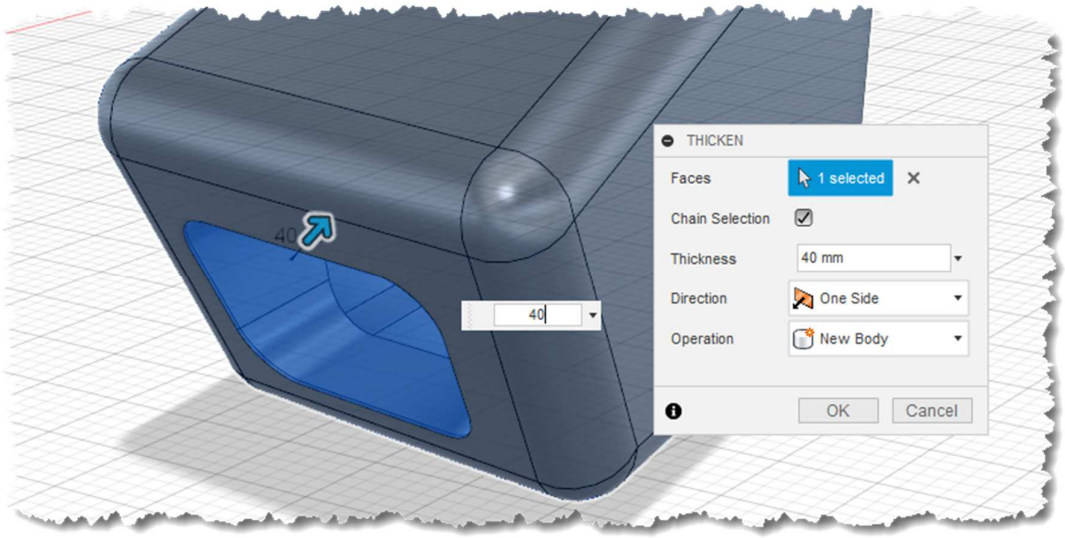




18. Use the “Stitch” command to join the internal and the external shells with the opening loft;



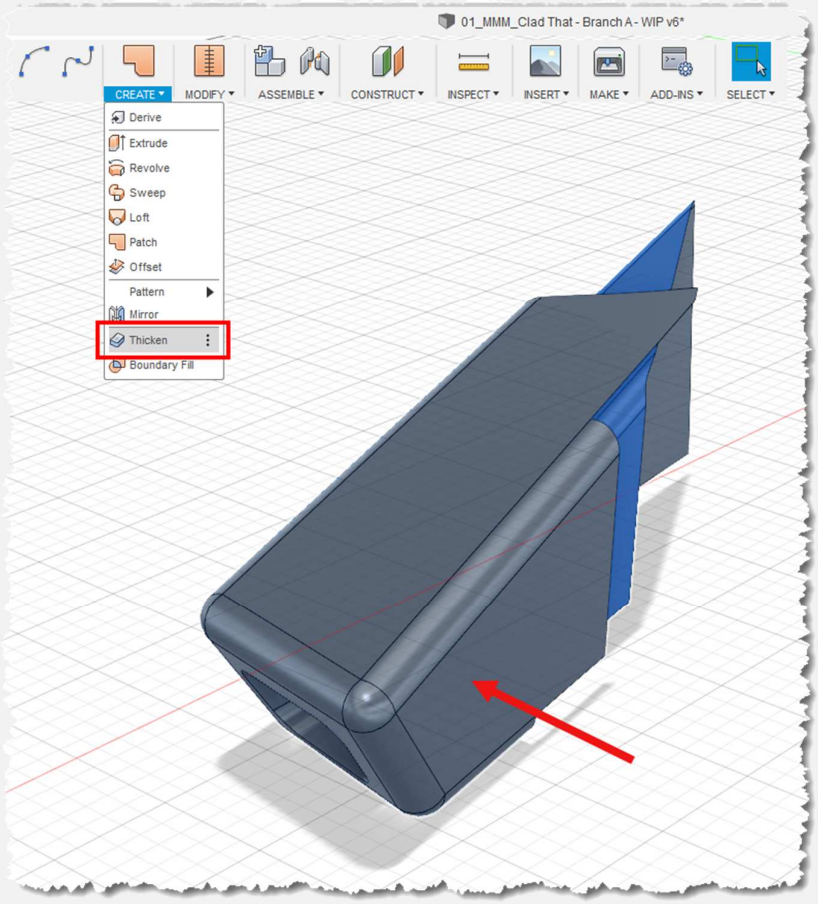
19. Select the result body and use the command “Thicken” to transform the polysurface to a solid element; Fusion 360 will create a new solid body without deleting the original surface;



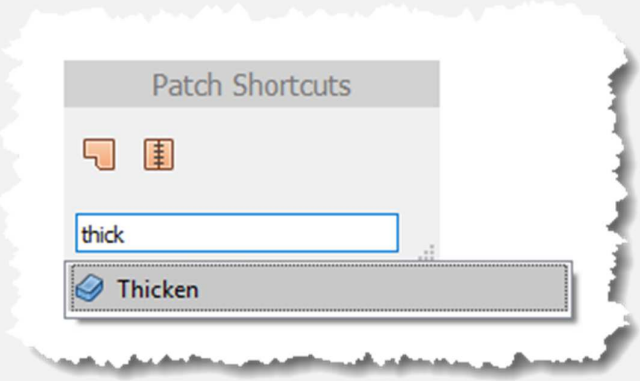
NOTE

The “Thicken” command can be called by selecting a surface, right click and by choosing “Thicken” form the contextual menu.

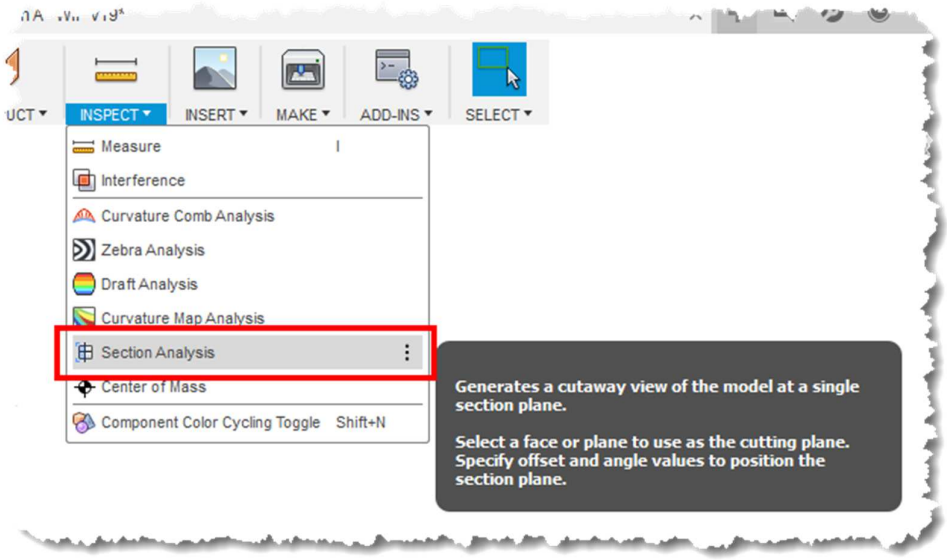
It can be also found under the “Create” menu of the “Patch” workspace.



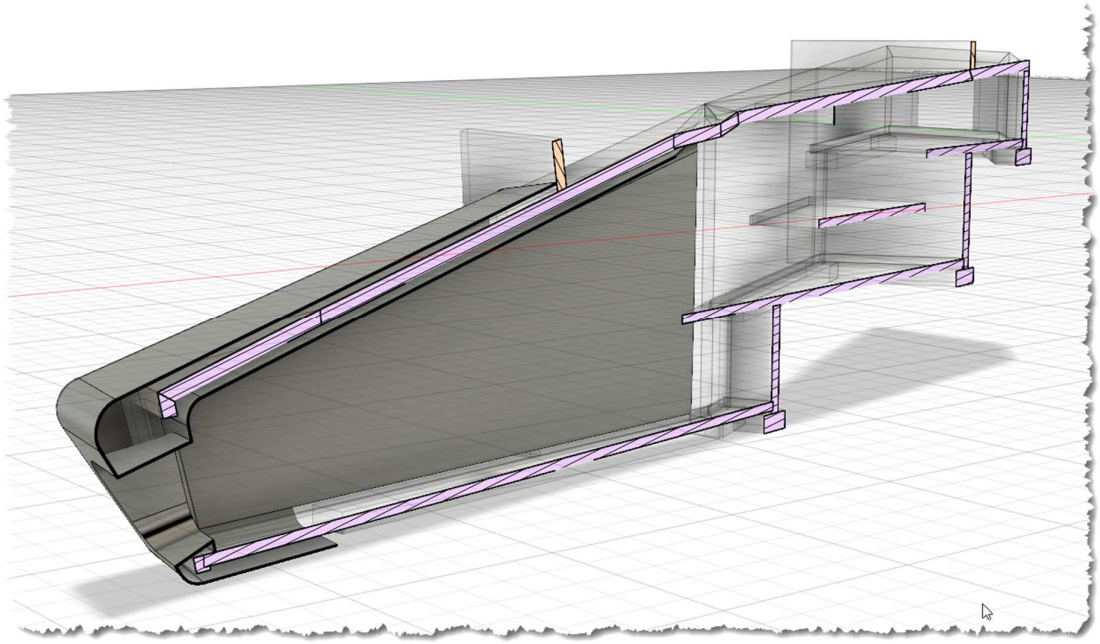
Another method is typing “S”; this opens the “Shortcuts” dialog. Here, start typing “Thicken”; Fusion 360 will then display the command by its name.

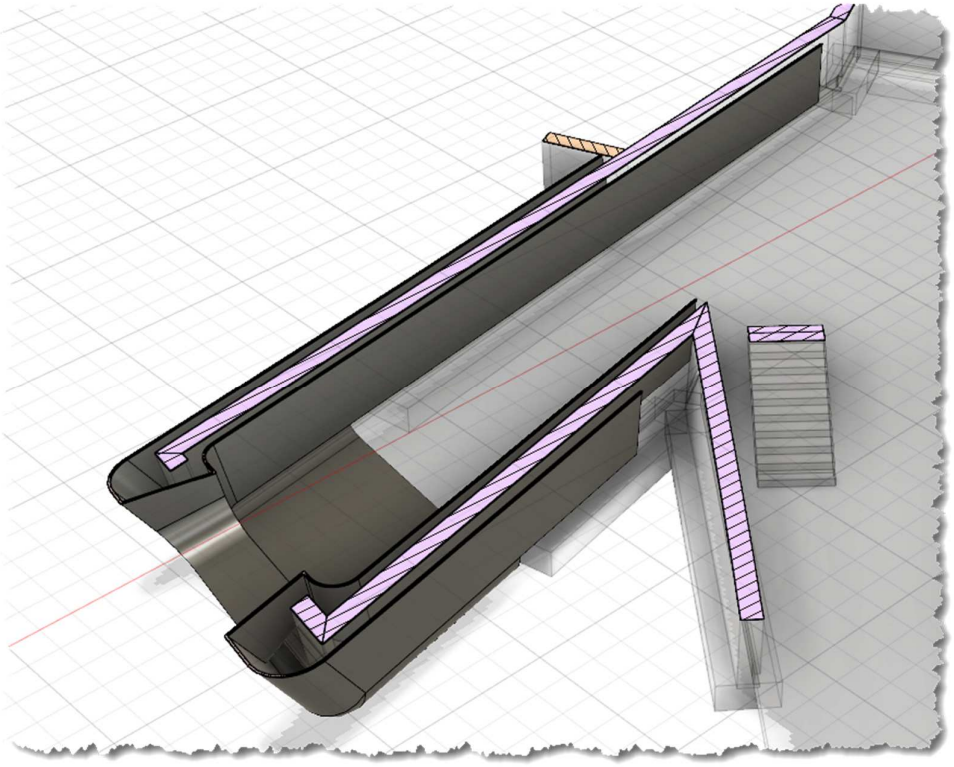


20. Under the “Inspect” menu select “Section Analysis”. This allows to select a face and then to generate a model section by just dragging the handle to define the placement of the section plane;

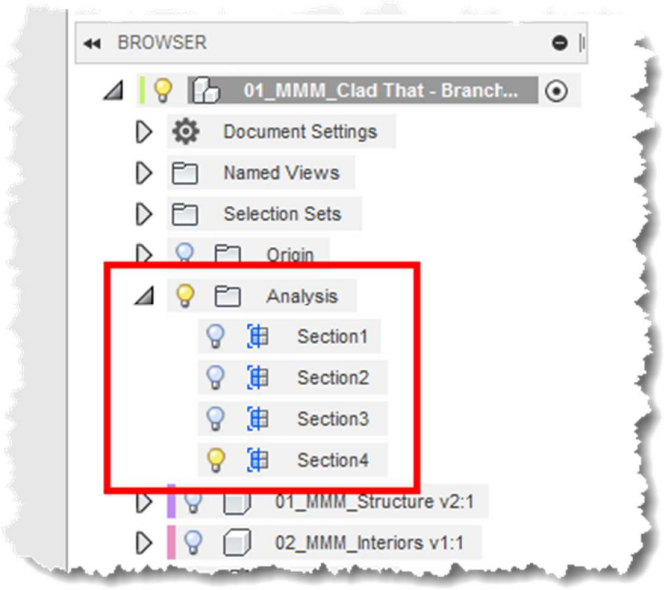


21. Following images show how the cladding shell is now a solid defined by its thickness;





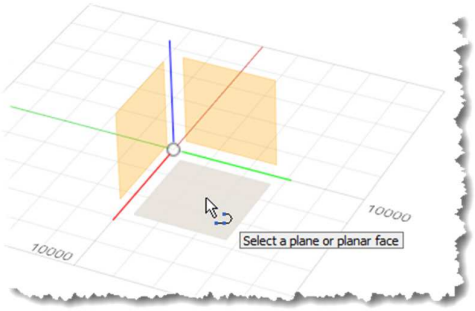
22. Use the specific group in the project tree to switch on or of the created cross sections;



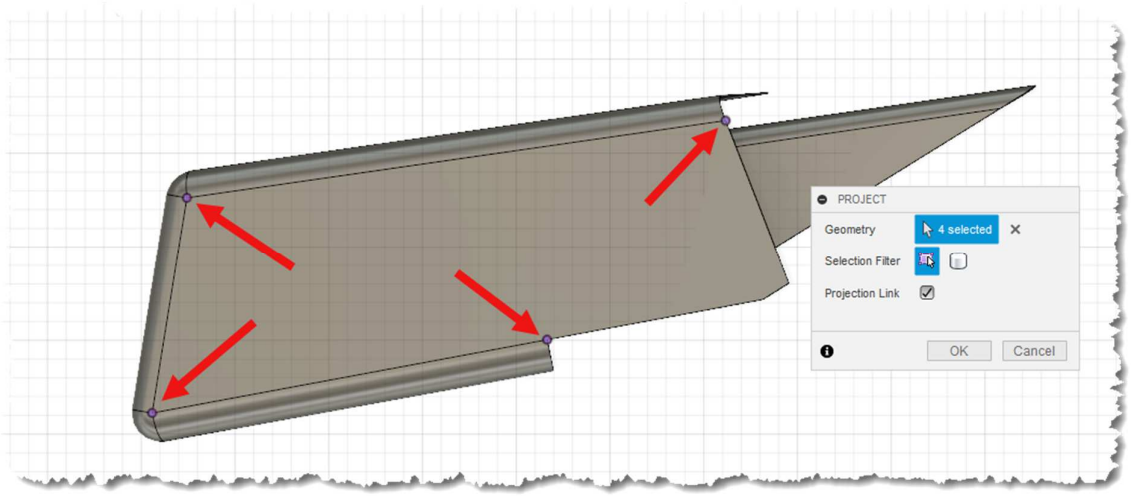
10.5 Panel Division

Once the shell has been modeled, it is necessary to divide it to define each single cladding panel. To do so, some sketches need to be defined; these will be then used to extrude a number of surfaces which will drive the shell division.

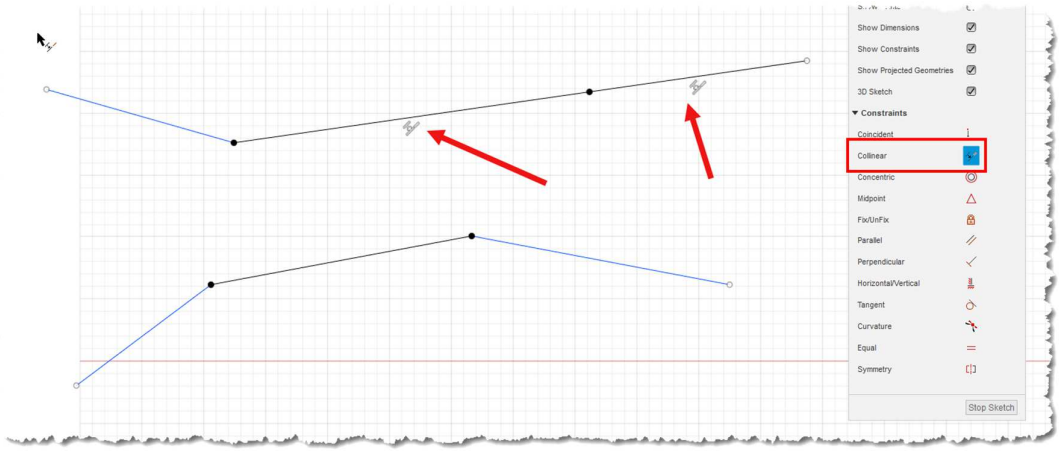
1. Start a new sketch using the XY plane as the sketch host;



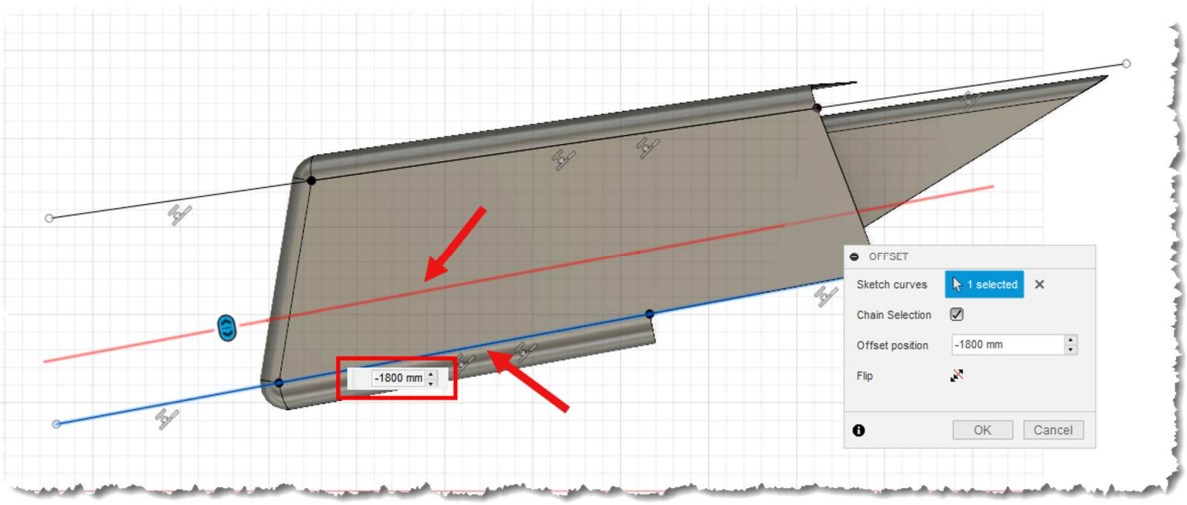
2. Use the “Project” command to select the four vertices identified below;



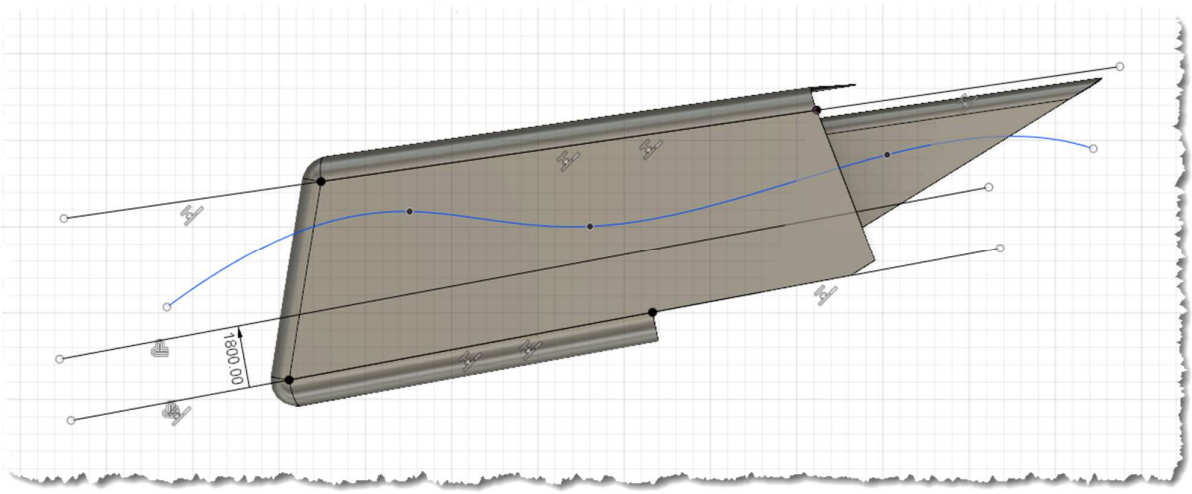
3. Sketch lines as shown on the next picture;



4. Use the “Collinear” constraint to align the “outer” lines with those having the projected vertices as endpoints;
5. Use the “Offset” command to create a parallel line with a distance of 1800 mm from the lower one;

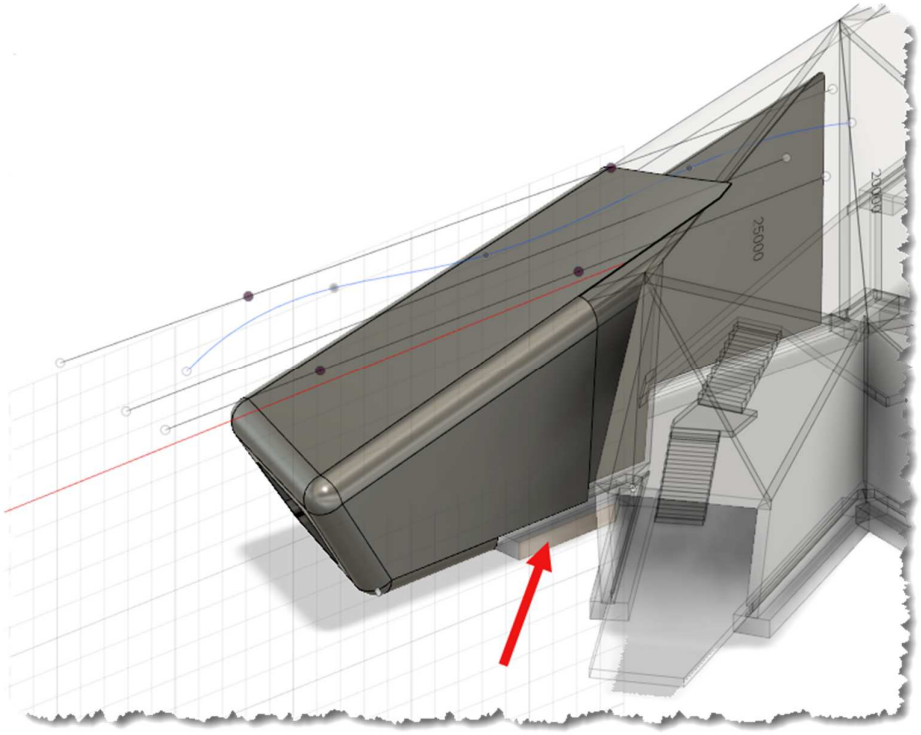


6. Insert a spline in the remaining space as shown below;

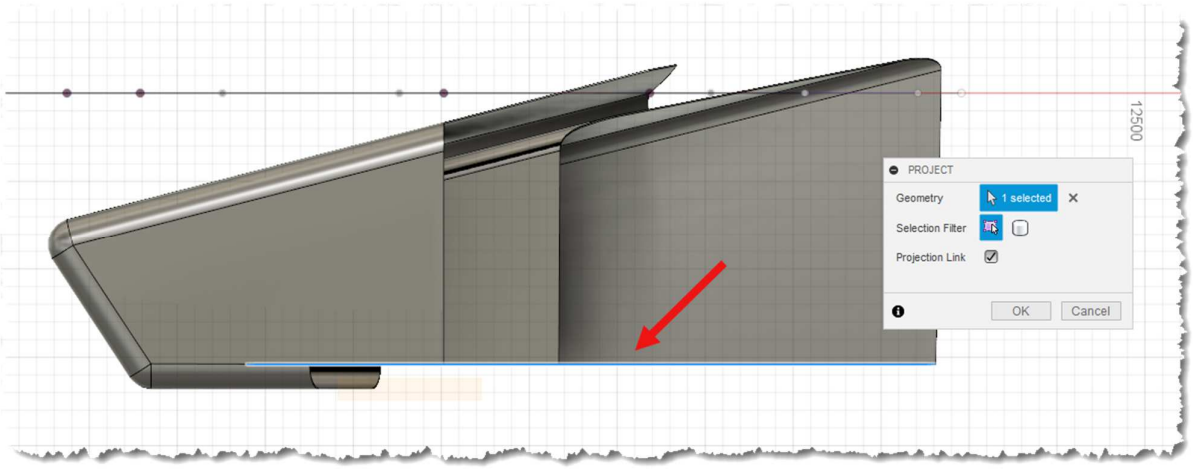


7. The sketch is finished; close it and rename it as “Vertical Cuts”;

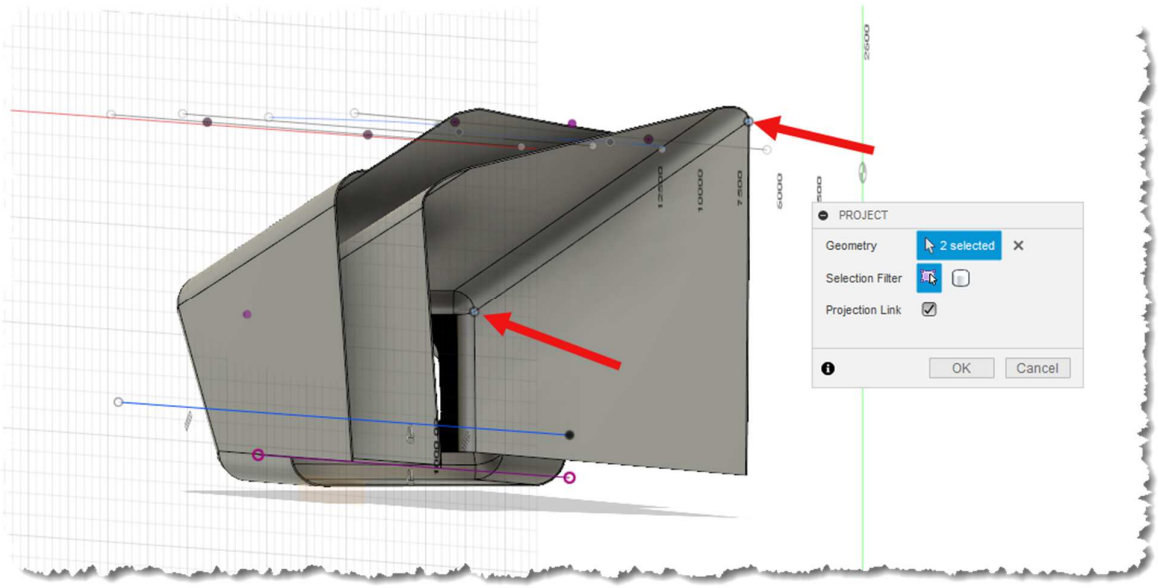
8. Start a new sketch using the vertical face of the continuous footing as the host;



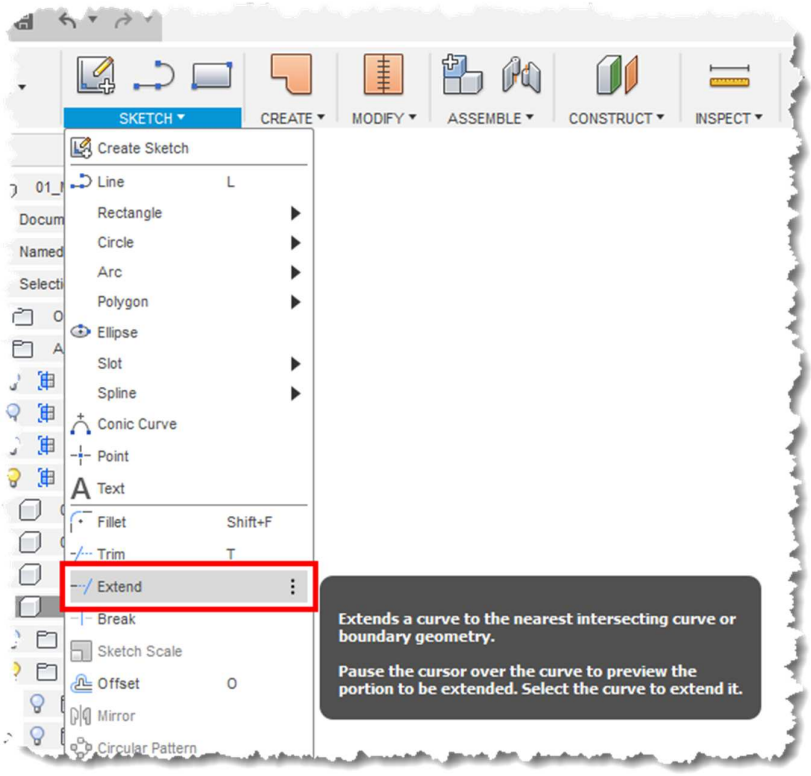
9. Use the “Project” command to capture the projection of the lower line; this is coplanar with the floor upper face; create an offset of this line upward and specify a distance of 1000 mm;



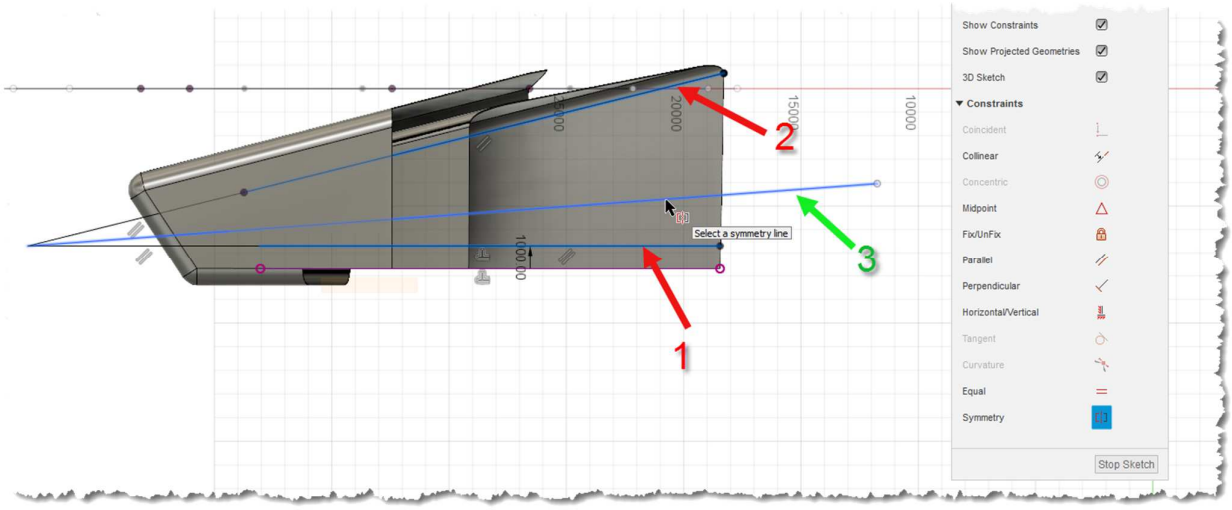
10. Use again the “Project” command to capture the two points identified on the next picture;



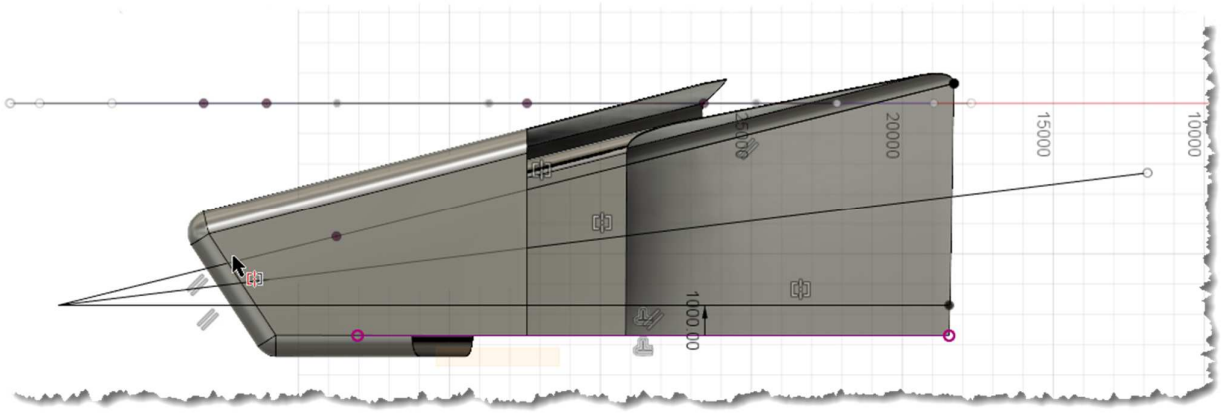
11. Create a line connecting these two points and add another collinear line to make the geometry to continue beside the shell shape;
12. Extend the slanted line and the horizontal one (that one created with the offset command) to find their intersection point;



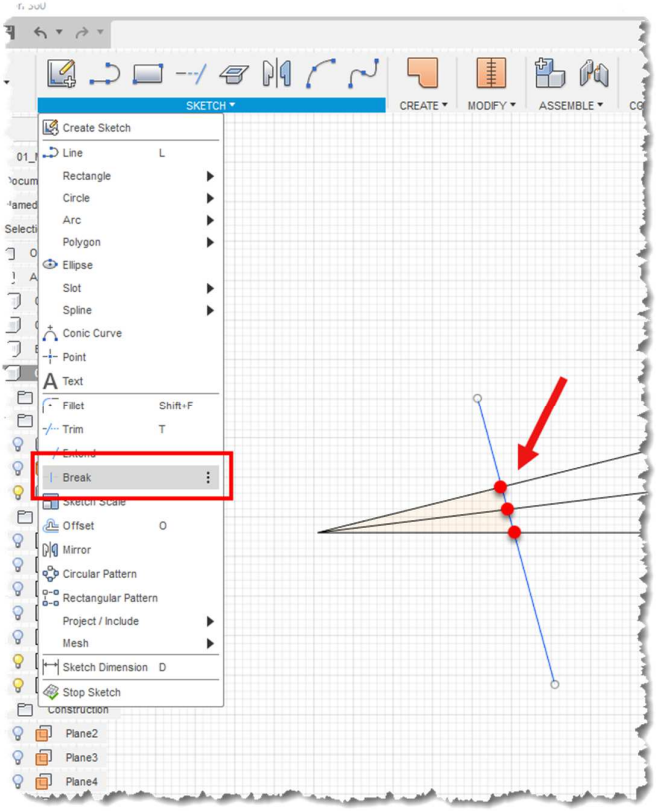
13. Sketch a third line between the other two ones;
14. Use the “Symmetry” constraint to force the third line to be the bisector of the angle;



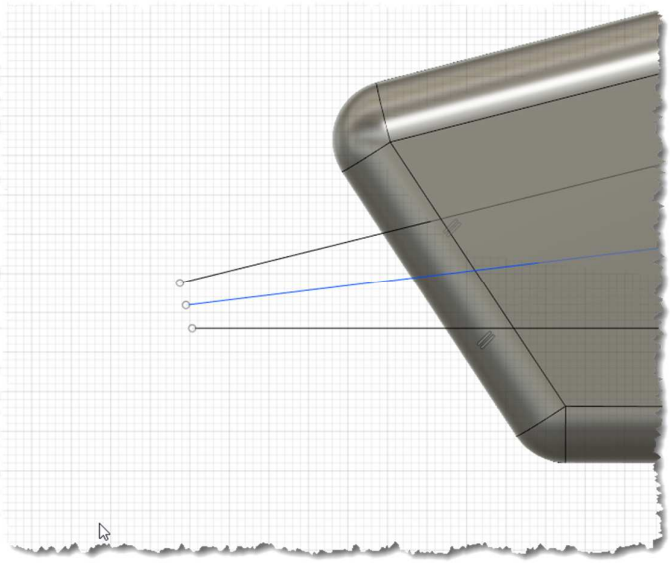
15. Lines should now be displayed in black since they are fully defined and constrained;



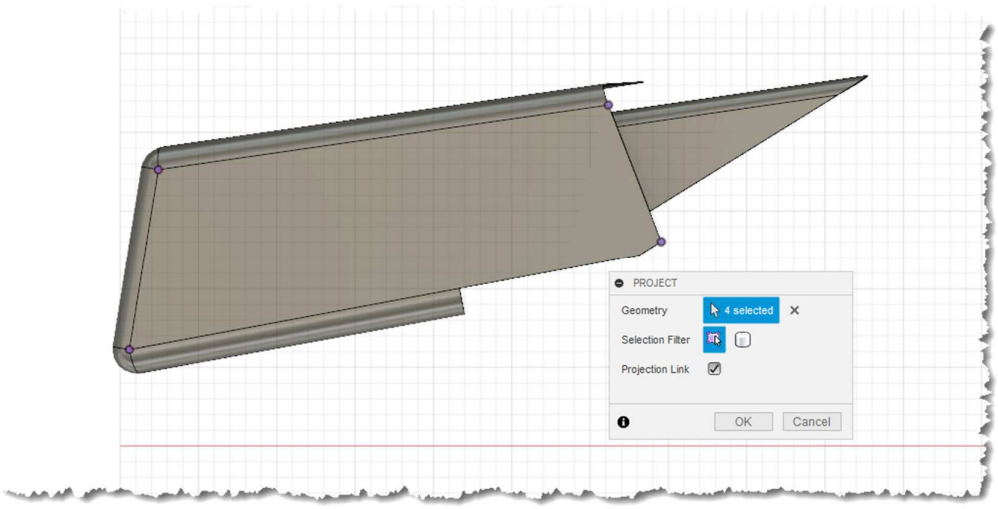
16. Create a line like that one in the picture below and use the “Break” command to split the three converging lines;



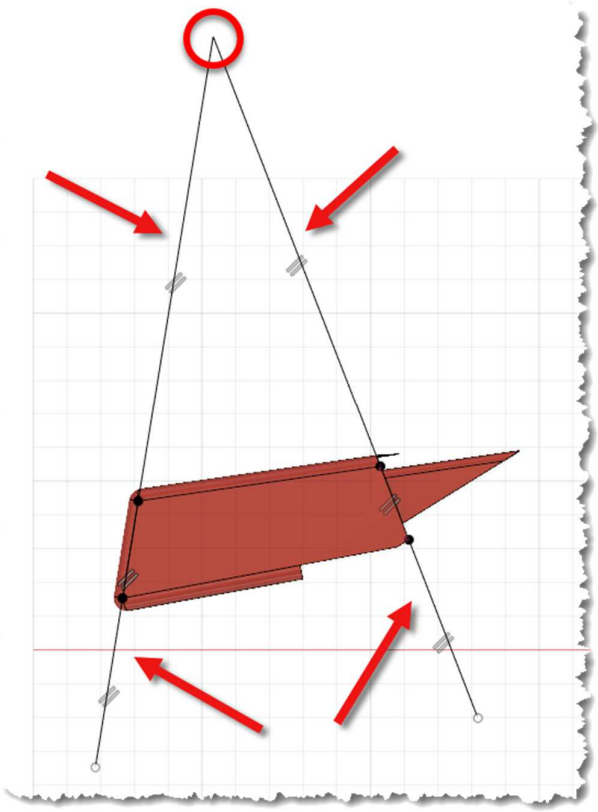
17. Remove the converging parts of the three lines. This will be useful later when the lines will be extruded. This way the result of the extrusion is three disconnected bodies while the lines with a point in common could have generated a connected body instead. An alternative is to select the converging portion of the lines and press “X”; this transforms the elements in construction lines which will not take part on the following extrusion;



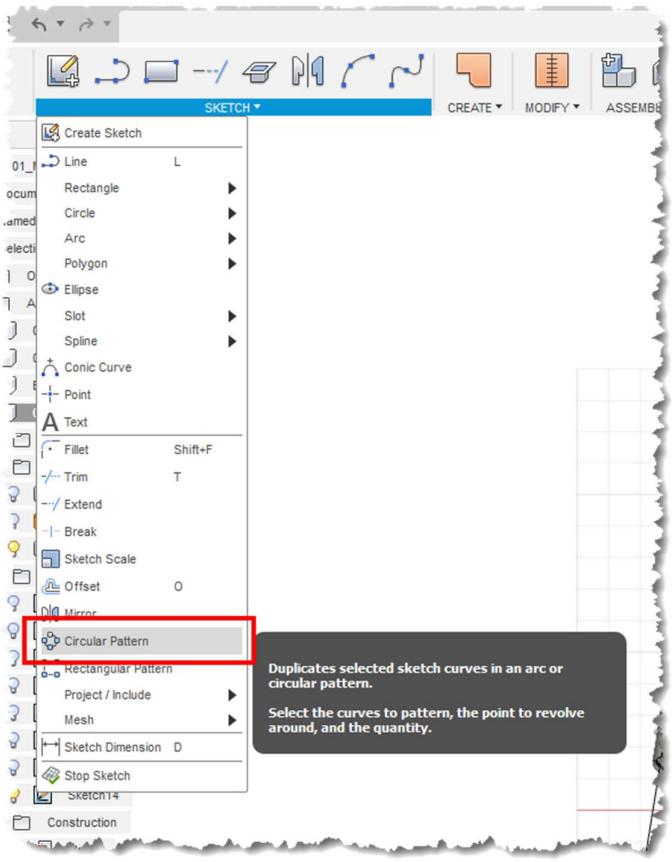
18. Finish the sketch and call it “Radial H Cuts”;
19. Create another sketch using the XY plane as the host;
20. Use the “Project” command to identify the same set of four points used before;



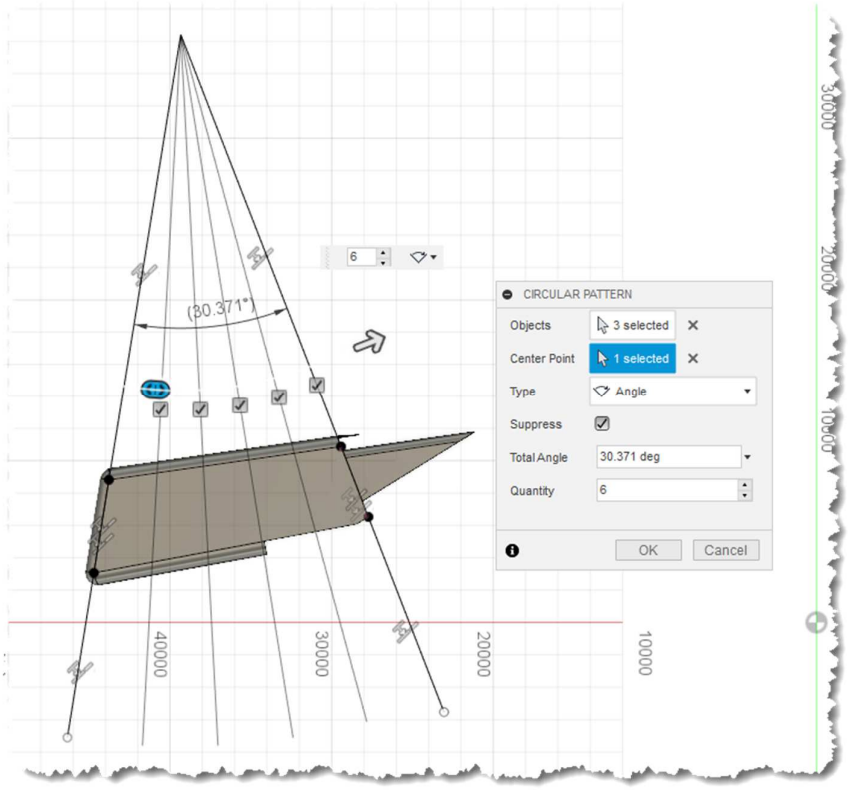
21. Create the lines connecting the first with the second couple of points and extend them (or add additional collinear lines) in order to define their intersection point. Also extend (or add additional collinear lines) the two lines below. The scope is also having the lines extending beside the body shape in order to drive an effective cut geometry;



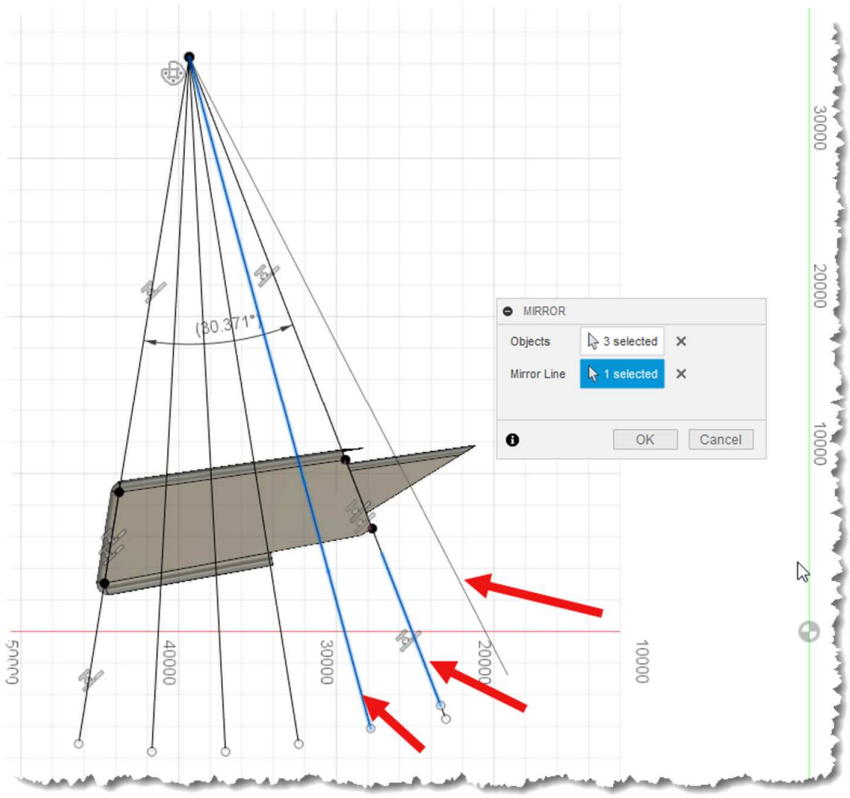
22. Use the "Circular Pattern" command to create a total of 6 radial lines starting from the intersection point;



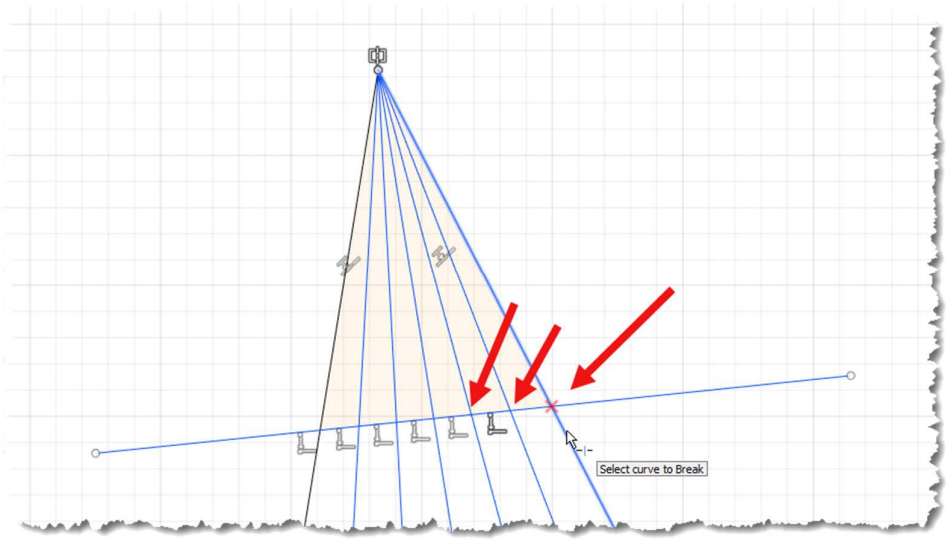
23. Specify the elements to be repeated, the center point, the number of items and the angle (this can also be measured before);



24. Mirror the second to last line in order to have an additional split element above the portion of internal shell on the right; use the last line as the mirror axis;

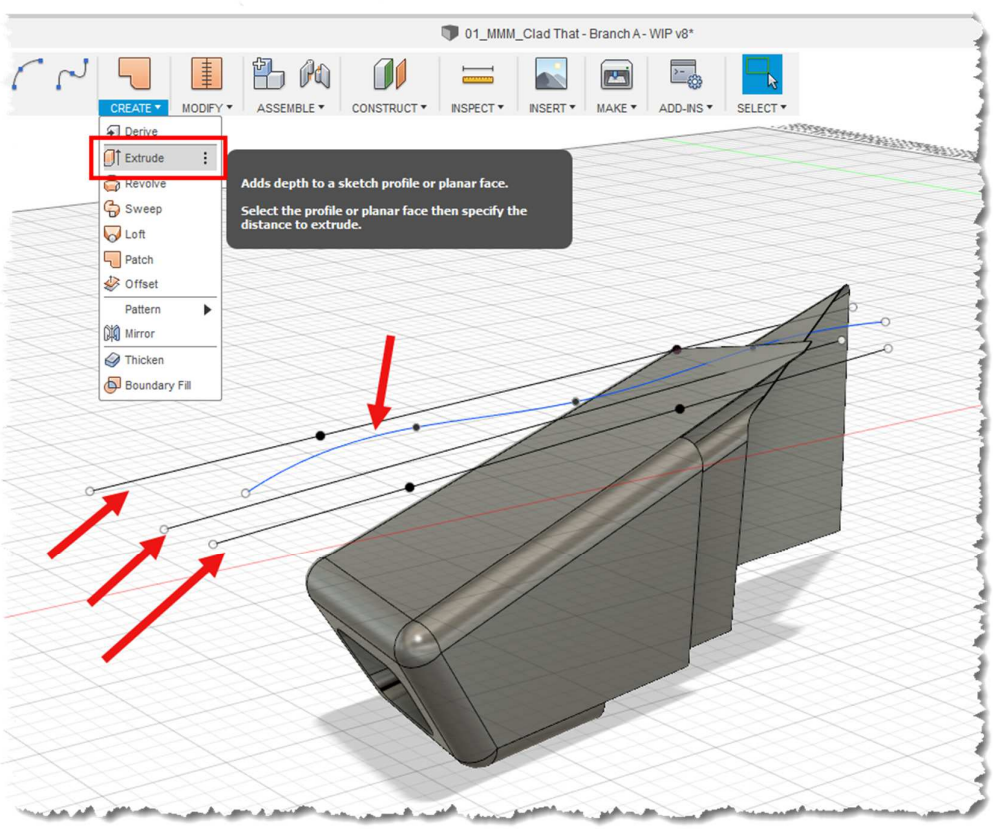


25. Cut the last part of the converging lines using the “Break” command as explained before;

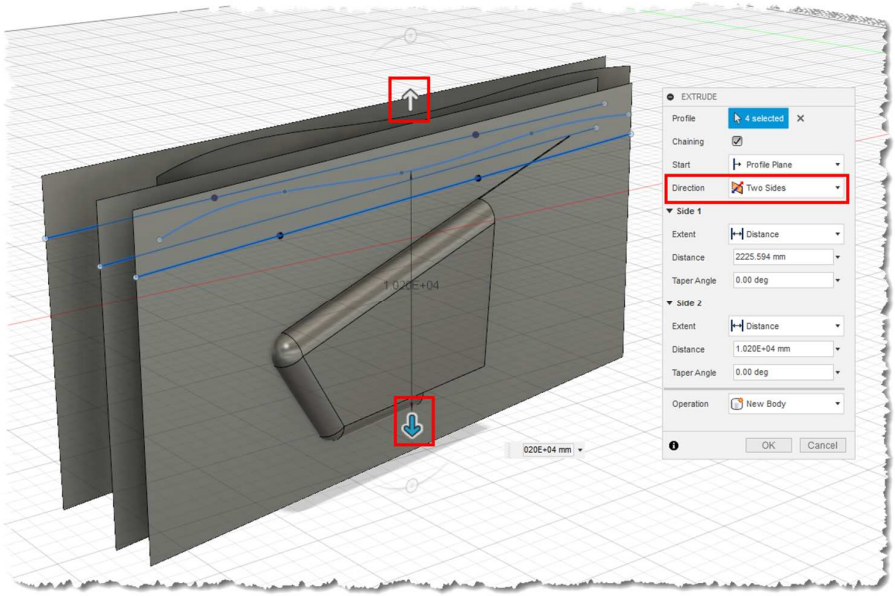


26. Finish the sketch and call it “Radial V Cuts”;

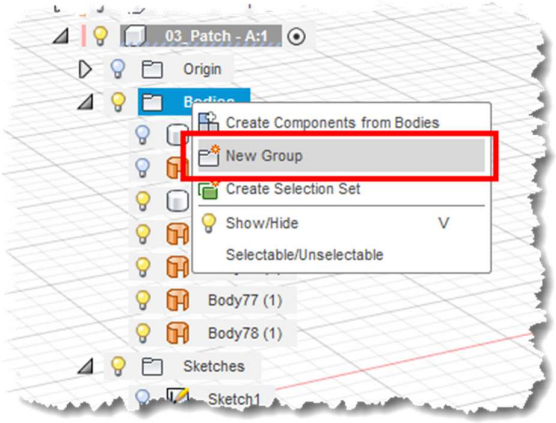
27. A set of three sketches has been created; these will be now used to create extruded planes to drive the panel division;
28. Turn on the visibility of the “Vertical Cuts” sketch and click on the “Extrude “command;



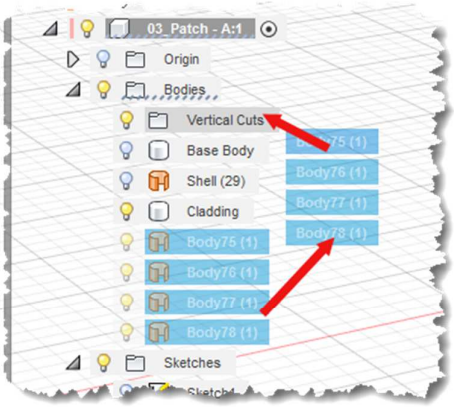
29. Select the four curves in the sketch and use, if needed, the “Two Sides” direction to make sure that the extrusion spans for the entire height of the cladding body;

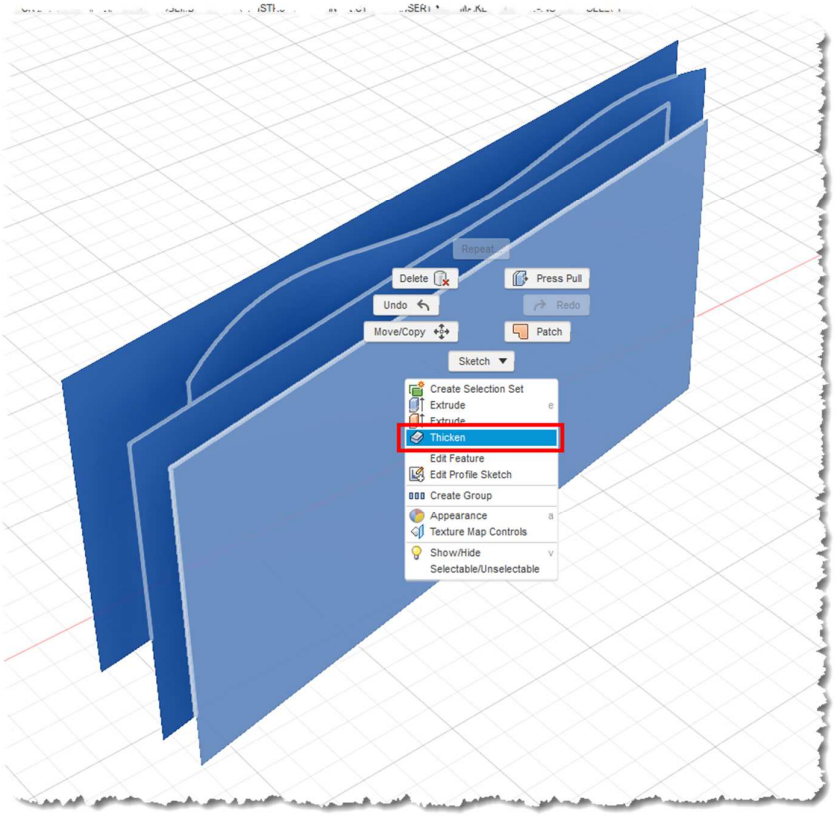


30. Under the Body section of the project tree, right click and create a new group called “Vertical Cuts”;

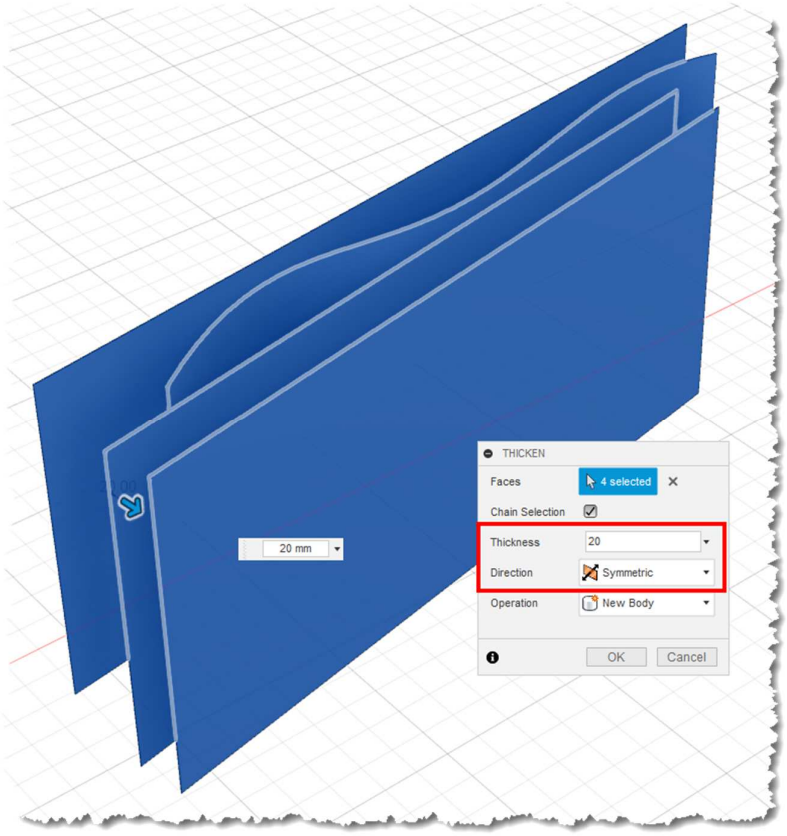


31. Drag the new bodies inside the group;
32. Use the “Thicken” command to define a 40 mm thickness for each of the created surfaces;

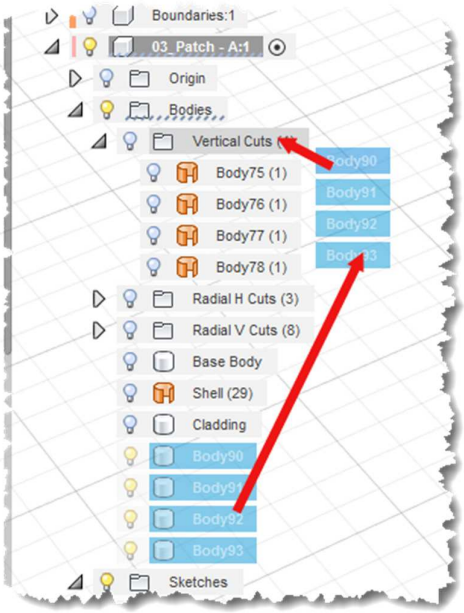




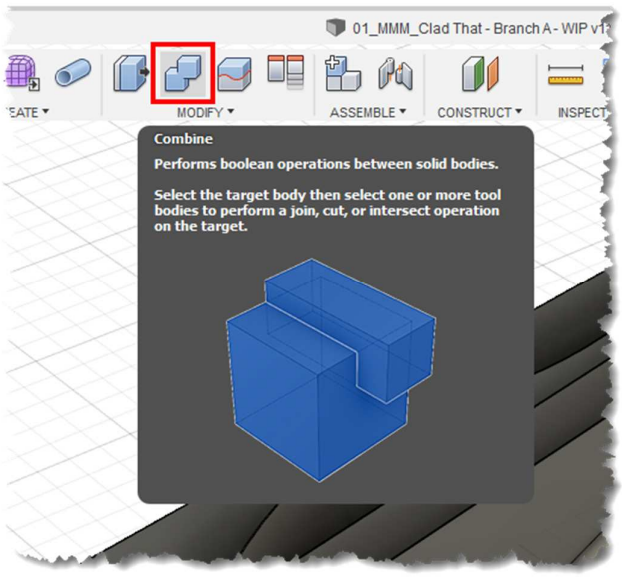
33. Specify “Symmetric” as the direction and 20 mm as the thickness (it means 20 mm for each side, for a total of 40 mm in this case);



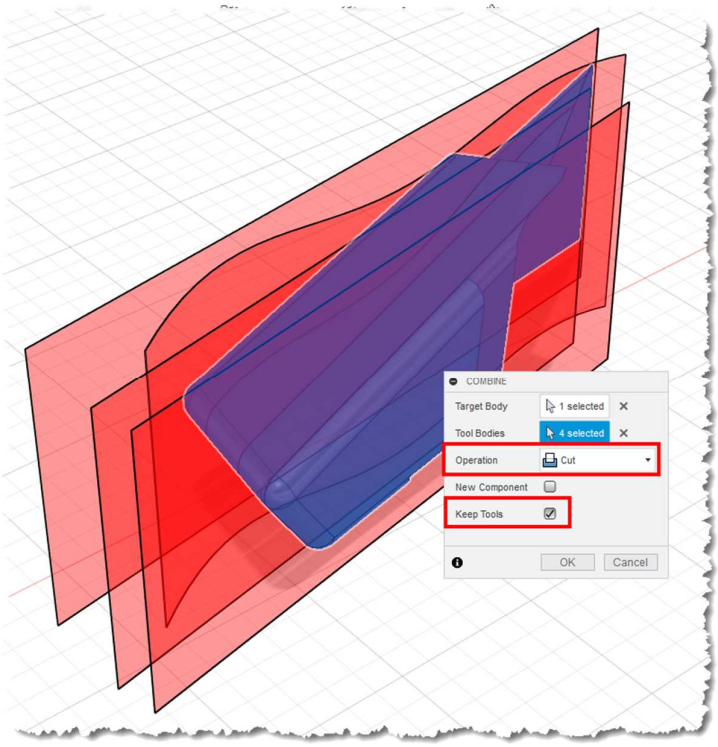
34. The thickness command generated new four bodies; drag them inside the Vertical Cuts group;



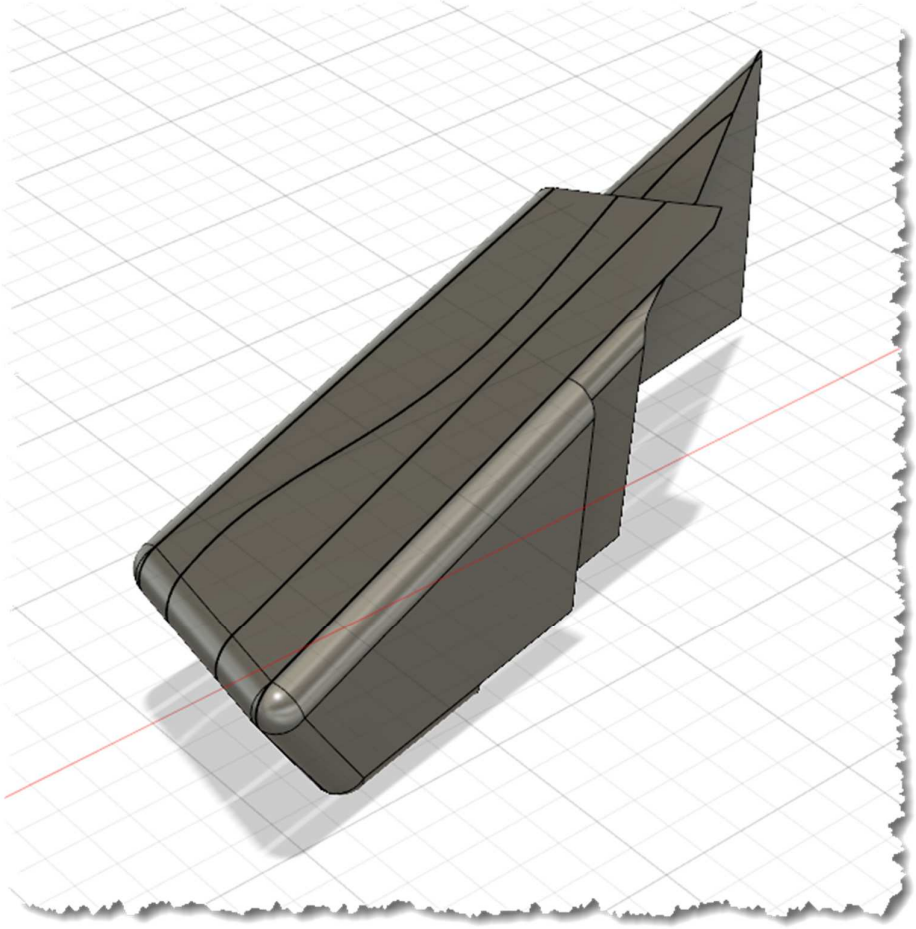
35. Change the workspace and choose “Model”;
36. Use the “Combine” command to cut the cladding shell with the vertical surfaces just created;



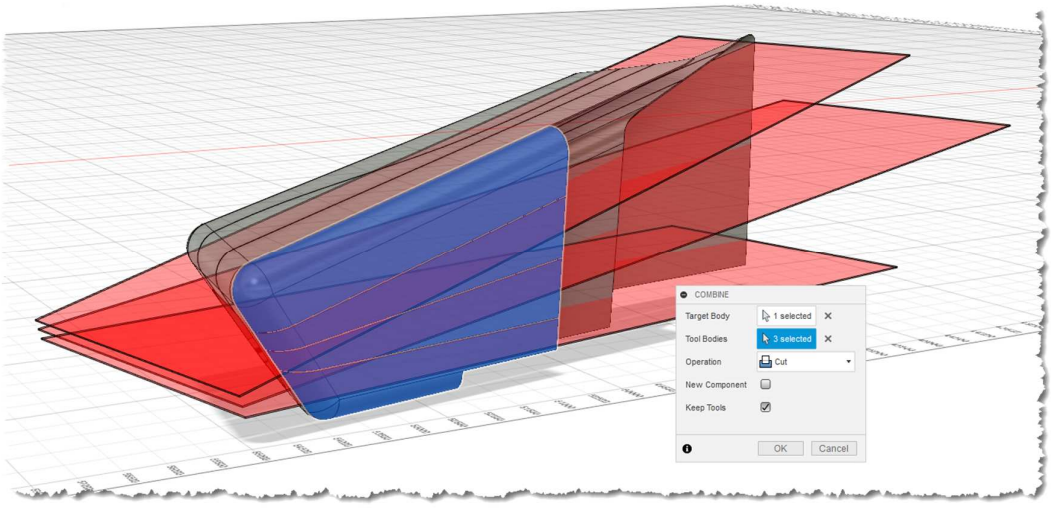
37. Select the shell as the target body and then the four vertical components as the tool bodies. The “Operation” must be set as “Cut” and the “Keep Tools” option must be checked to maintain the vertical elements and be able to re-use them in a second moment. Click “Ok” when done;



38. The result should be like that one shown below;

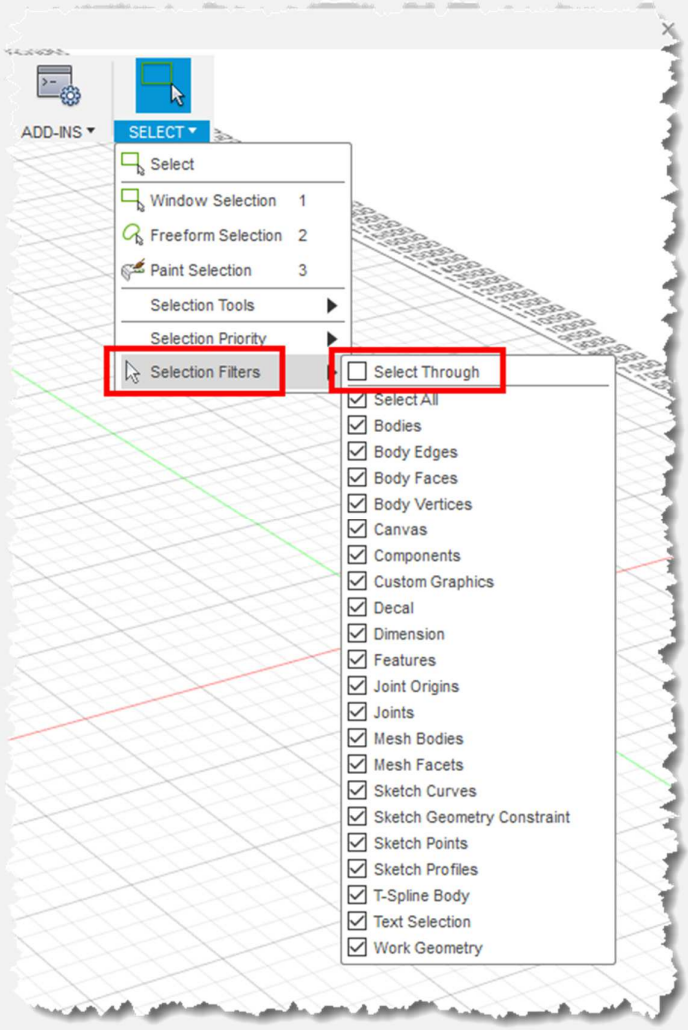


39. Repeat the same process for the other sketches;



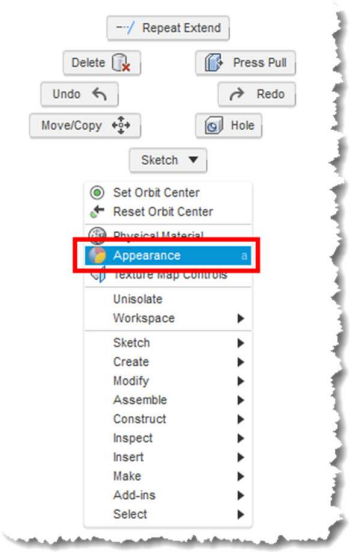
NOTE

In some cases, selecting elements can be difficult since Fusion 360 could give the priority to components “behind” that to be selected. In this case, disable the “Select Trough” option.

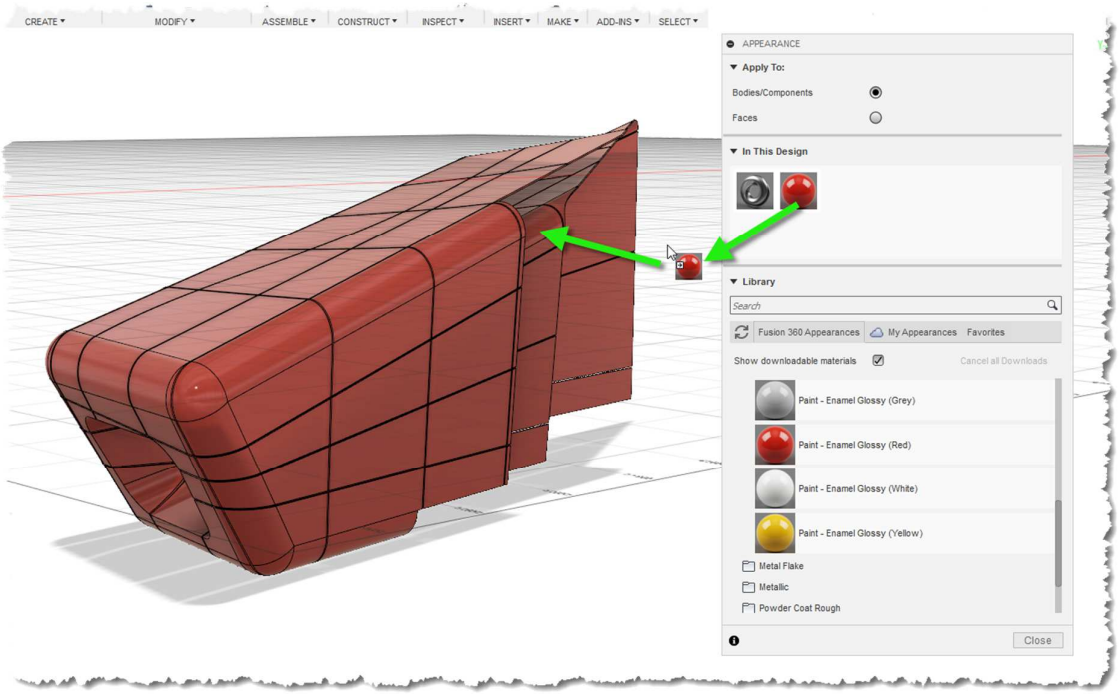


This also allows to set a selection filer, making possible to reduce the set of selectable elements or to force the selection to a specific element only.

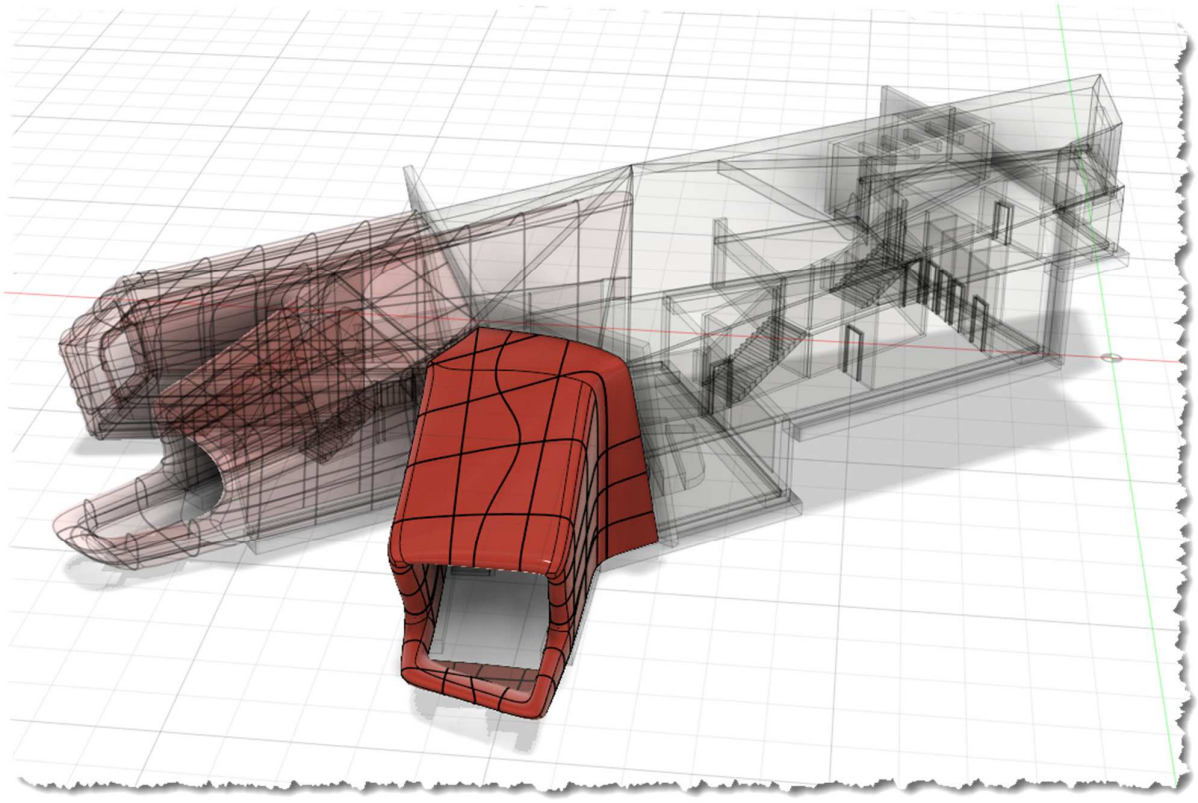
40. Use the “Appearance” dialog (right click on the work area and choose the “Appearance” command) to define new material or to load them from the Fusion 360 library;



41. Assign the material to the desired bodies by dragging and dropping the material icon above the desired elements;



11 Exercise #3: The Freeform Environment – one Branch and one Theatre



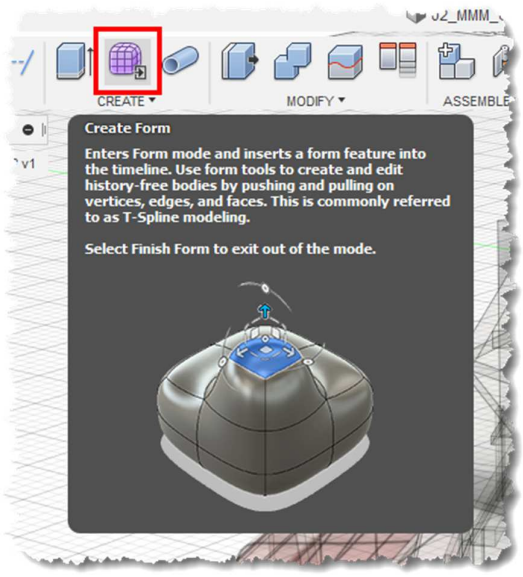
The scope of the current exercise is understanding the main modeling techniques inside the freeform/sculpt environment. This allows to freely manipulate facets, edges or vertices using the T-Spline technology.

Use the file “02_MMM_Clad That - Branch C.F3D” and start a new design with it; otherwise open the project from the A360 folder.

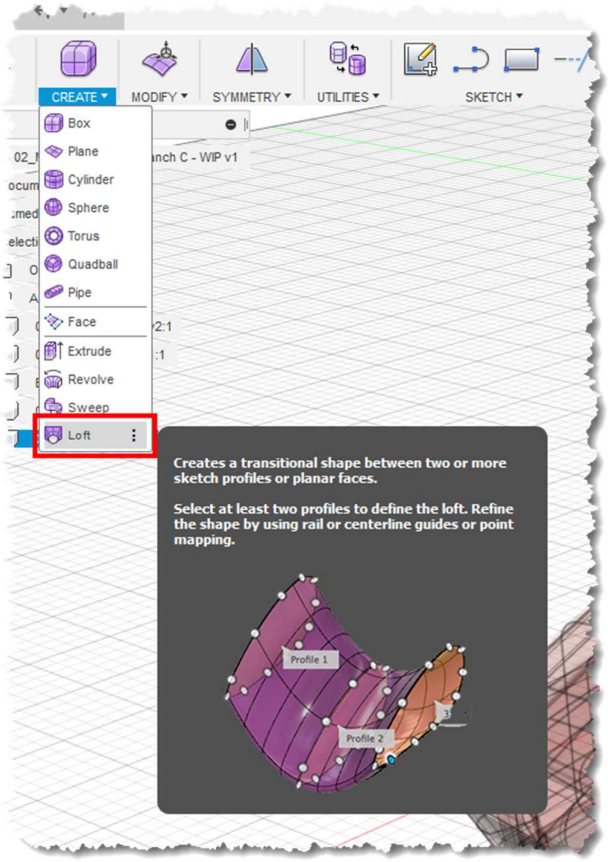
Create a new component called “04_Freeform – C” (“C” stands for the building branch) and activate it.

11.1 Base Geometry Creation

1. Create a new component and call it “2_Extrusion”; make it active;
2. Click on the “Sculpt” icon to activate the freeform workspace;

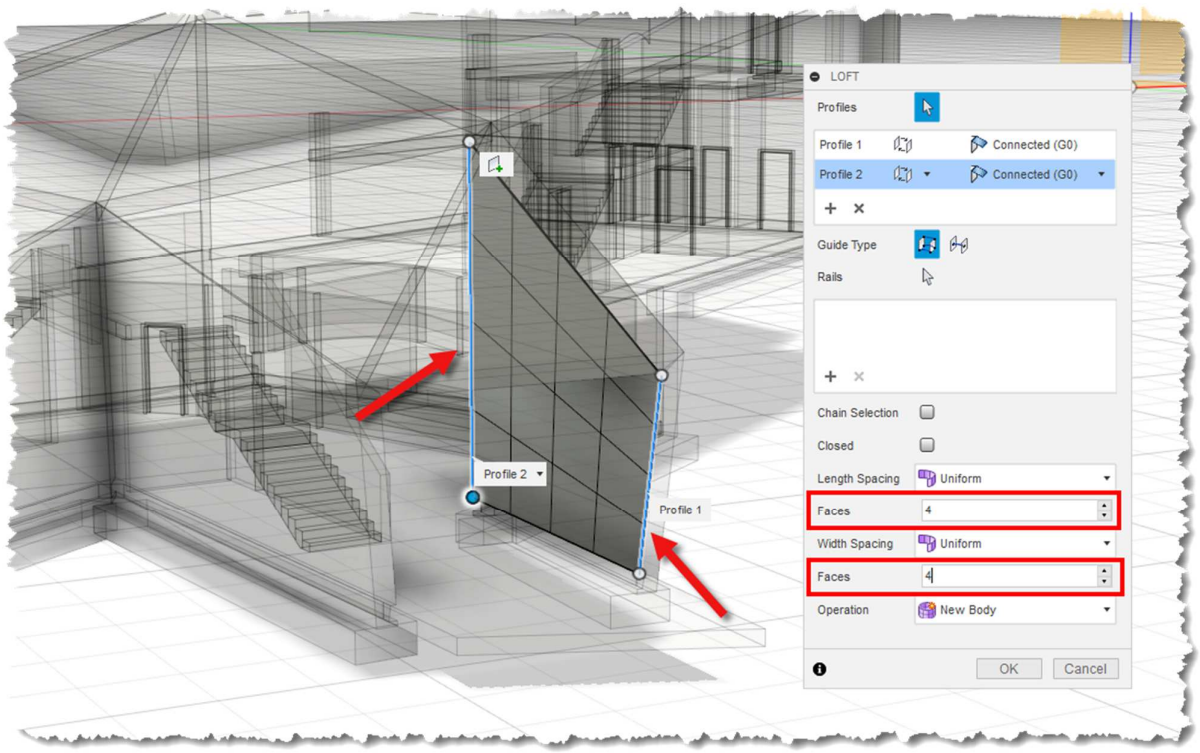


3. As in the previous exercise, the “Loft” command is going to be used here, too. In this case, the command will generate a T-Spline geometry instead of a B-Rep one;

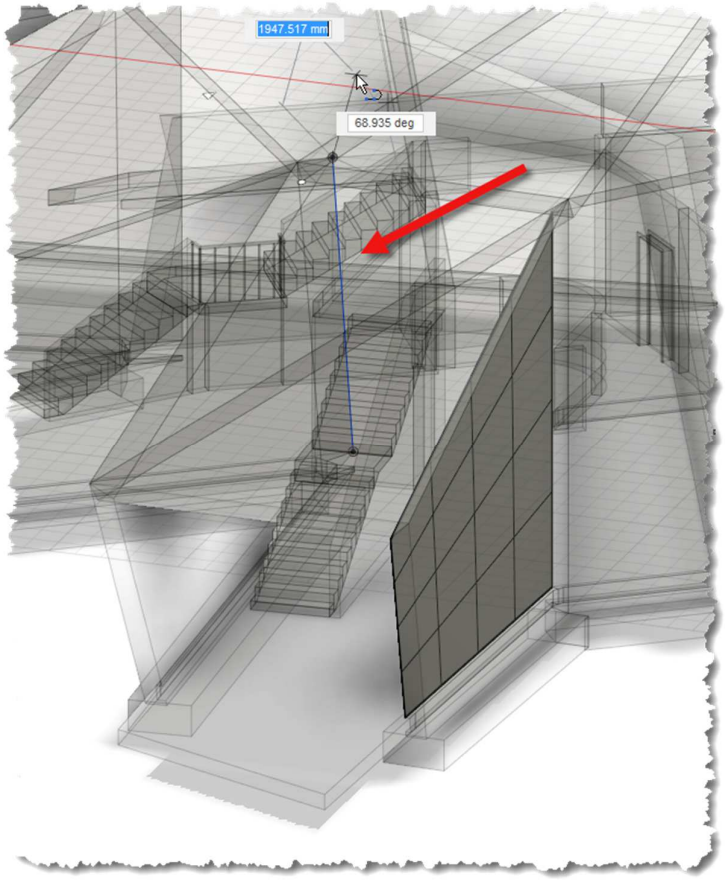


NOTE
A T-spline surface can be thought of as a NURBS surface for which a row of control points is allowed to terminate without traversing the entire surface. The control net at a terminated row resembles the letter "T". Modeling surfaces with T-splines can reduce the number of control points in comparison to NURBS surfaces and make pieces easier to merge but increases the book-keeping effort to keep track of the irregular connectivity. T-splines can be converted into NURBS surfaces, by knot insertion, and NURBS can be represented as T-splines without T's or by removing knots. T-splines can therefore, in theory, do everything that NURBS can do. In practice, enormous amount of programming was required to make NURBS work as well as they do and creating the equivalent T-Spline functionality would require similar effort.

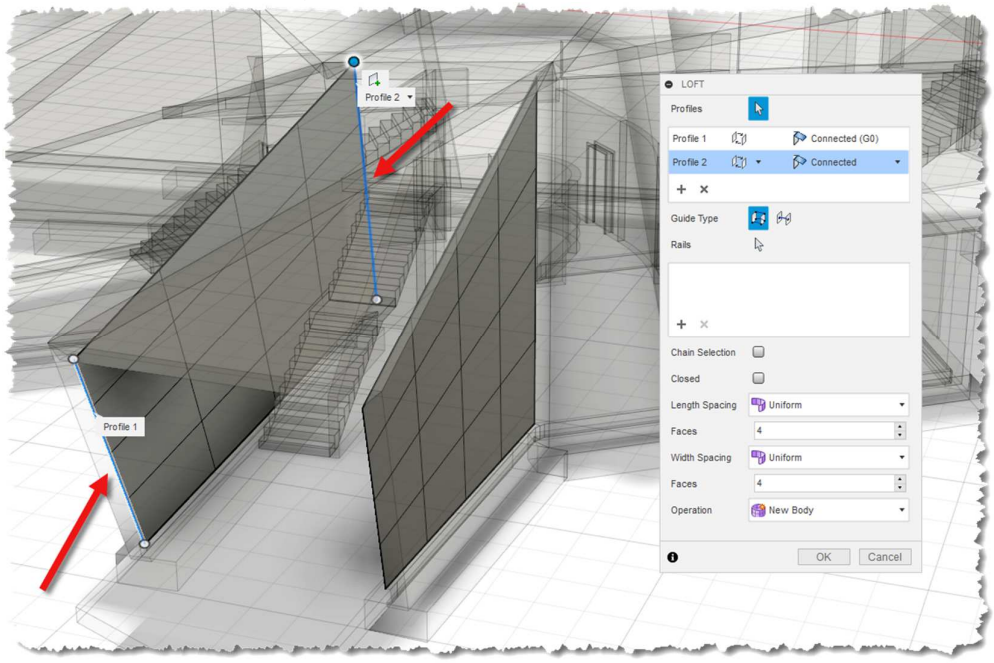
4. Select the internal edges as shown in the next picture and specify 4 faces both for the Length and for the Width directions. Avoid dividing the surface with too many faces since it will become more difficult to edit the shape and the resulting geometry can lose smoothness or have discontinuity areas;



5. As in the previous exercise, for the second vertical surface, it is necessary to create a sketch to define a continuous edge line since the wall edge is interrupted by an intersecting floor;



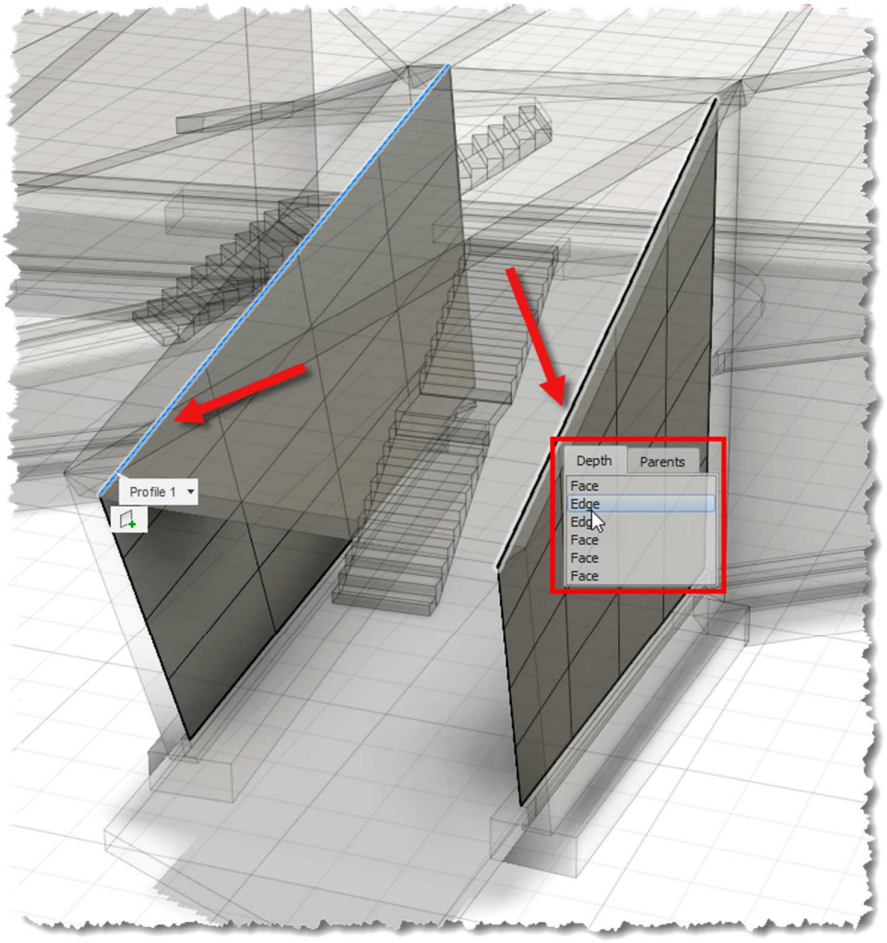
6. Create another loft using the same approach;



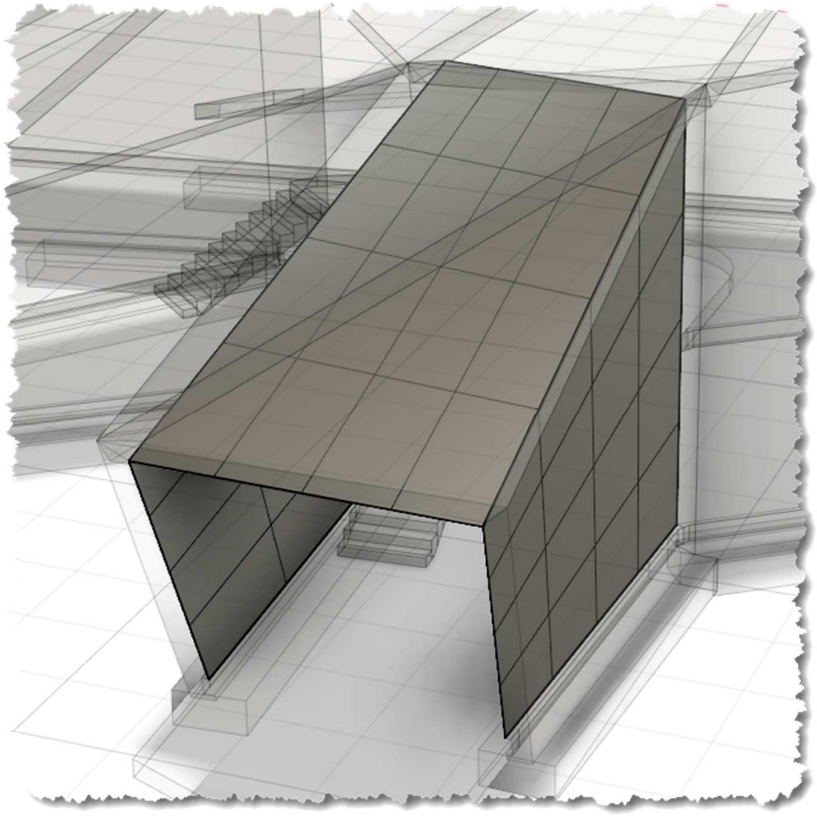
7. It is possible to proceed in two different ways to create the “ceiling” of this internal shell:
    a. Adding another loft element and joining its edges with the two vertical walls’ top edges;
    b. Creating a “bridge” between the two vertical walls;

11.1.1 Loft and merge

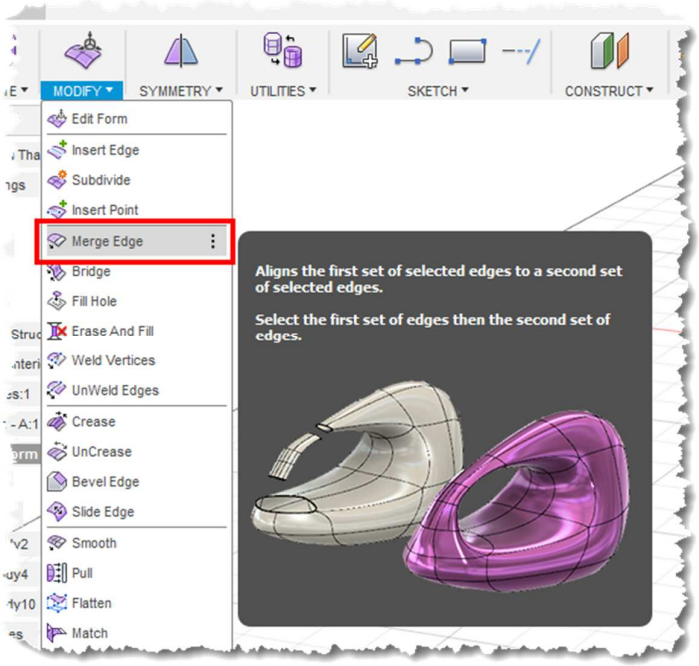
1. Create a new loft using the edges or the roof; if selecting an edge is difficult, keep the left mouse button pressed until a dialog appears: this allows to choose between all the overlapped and selectable elements;



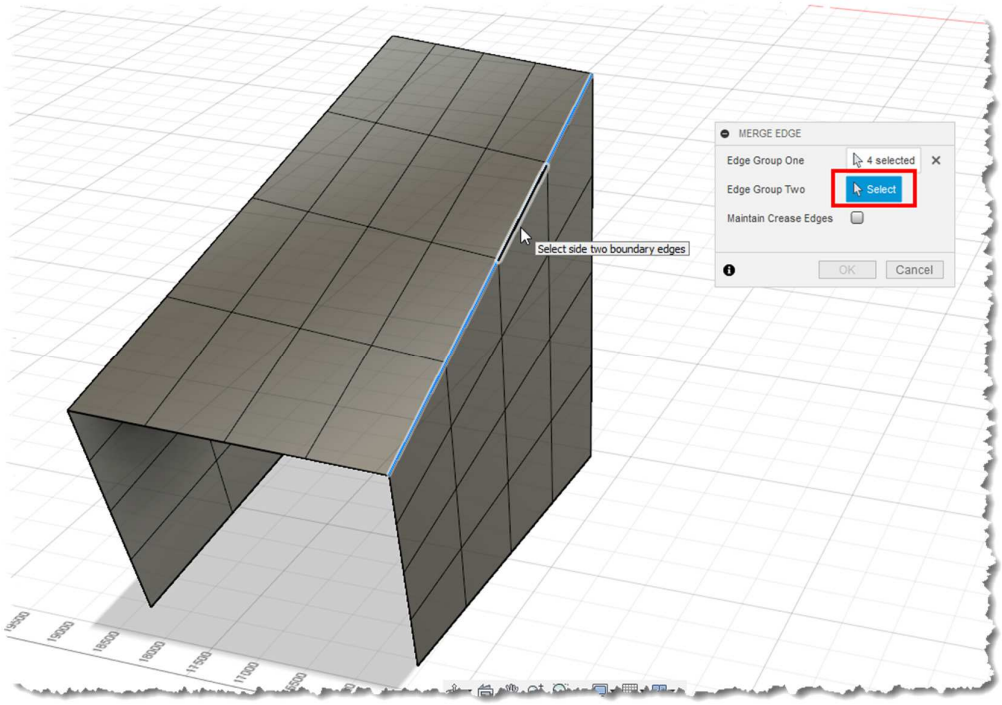
2. Specify 4 faces both for Length and for Width directions and click “Ok”;



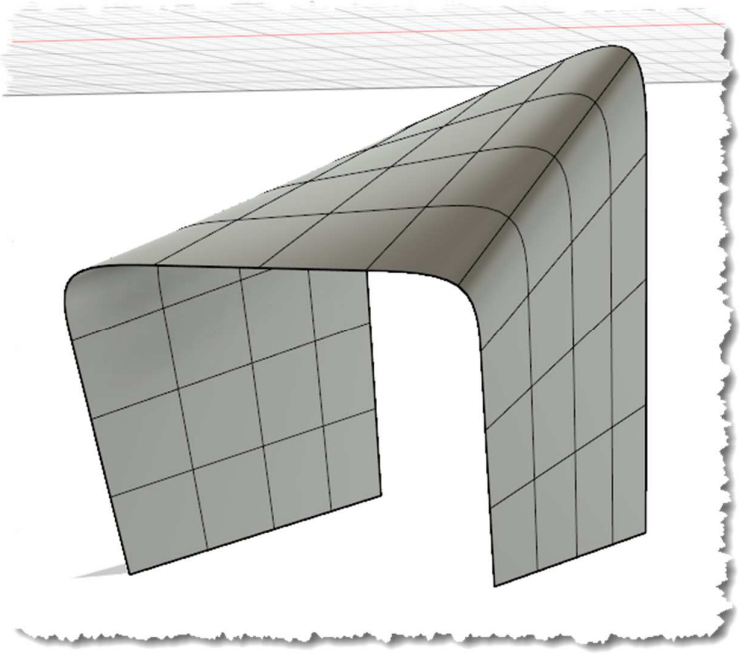
3. Turn off the visibility of the other components;
4. Click on the “Merge Edge” command; this allows to select a group of edges and to merge them with a second group to create a continuous surface;



5. In this case the two groups of edges are almost precisely overlapped: double click for selecting the first sequence composed of 4 segments; click con the “Edge Group Two” to enter the second group and double click again on the edge. Fusion 360 understands that a first group of segments have been selected so it keeps as selectable the edges that have not been selected yet only; this method allows to quickly select the two groups avoiding mistakes;

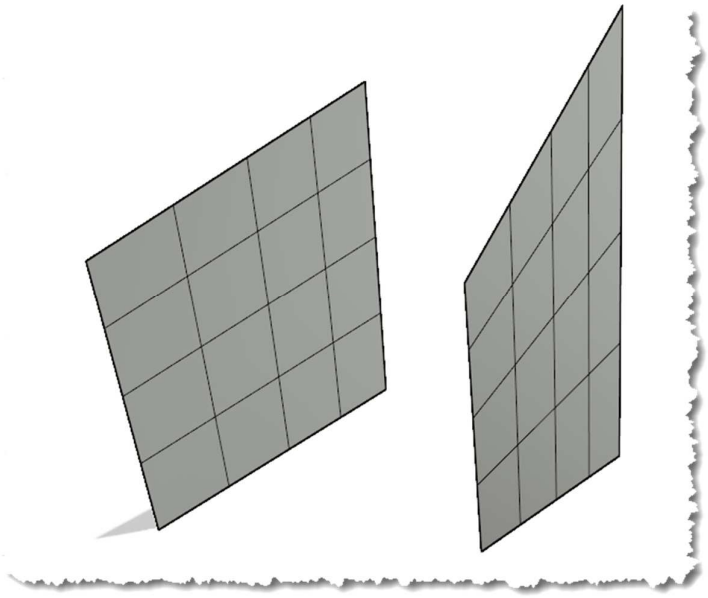


6. Click “Ok” when done and repeat the same procedure for the other edge;

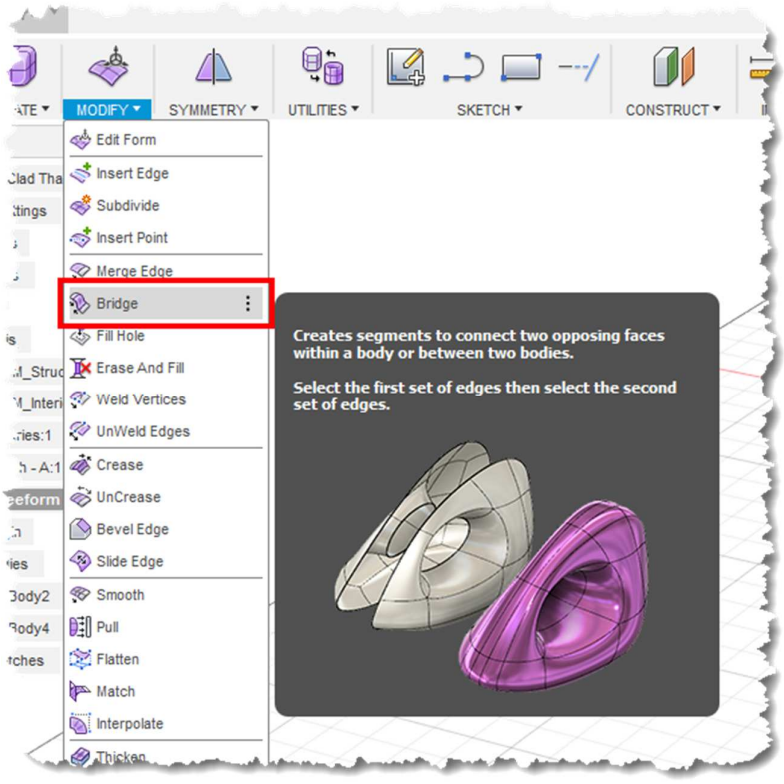


11.1.2 Bridge creation

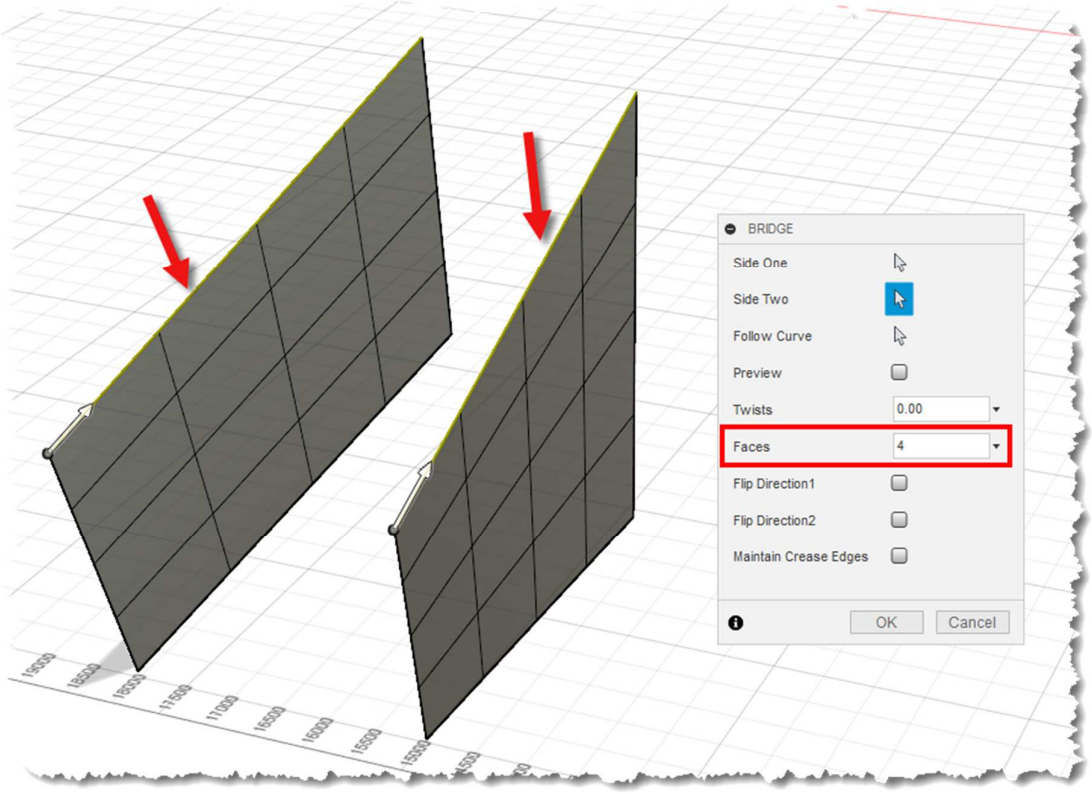
1. Undo the “Merge Edge” command and the creation of the “roof” surface to come back to the elements as in the next image;



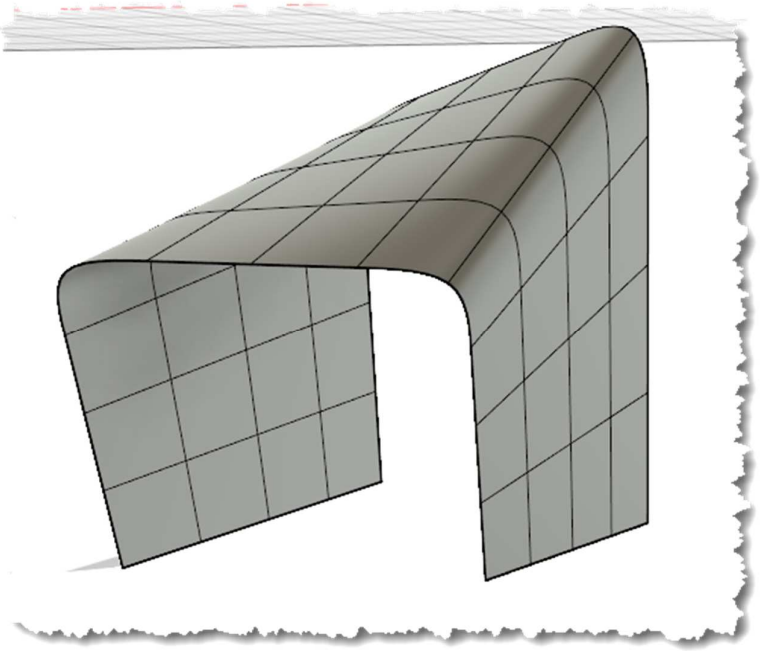
2. Click on the “Bridge” command under the “Modify” menu of the “Sculpt” environment;



3. This command requires to select two edges composed of the same number of segments and to specify how many faces must be created to interpolate the first edge with the second one. In this case 4 faces will be set;

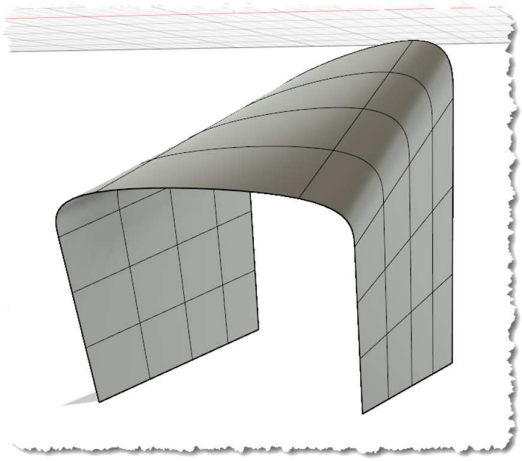


4. Click “Ok” when done and compare the result with the previous method;

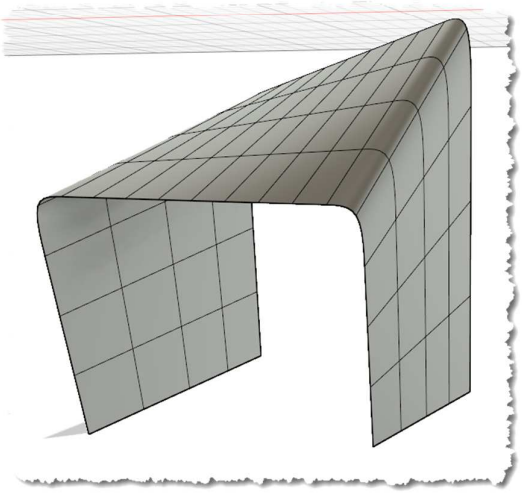


5. Try to set a different number of faces and check the differences on the created geometry;

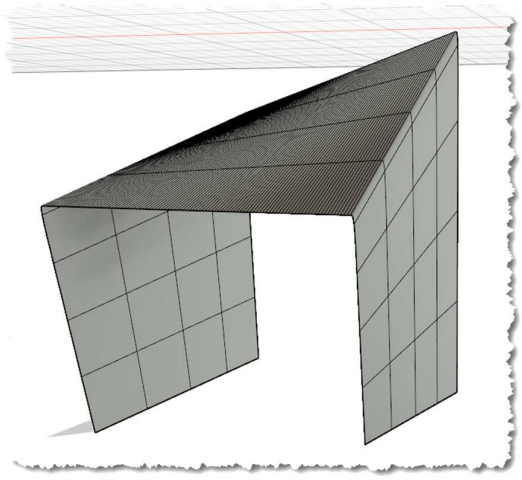
1 face



10 faces



100 faces

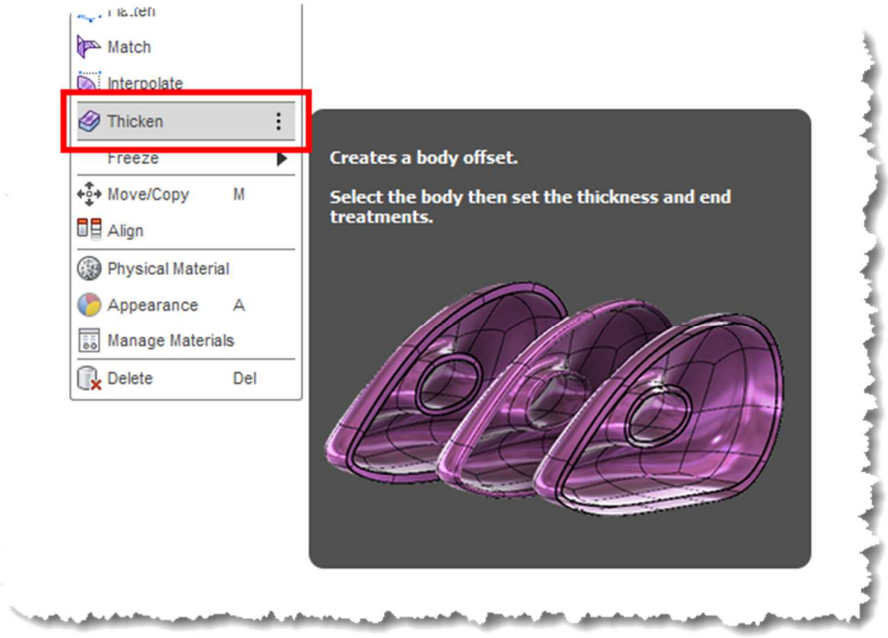


The base geometry is now created; the next step is using it to model the internal and the external shells.

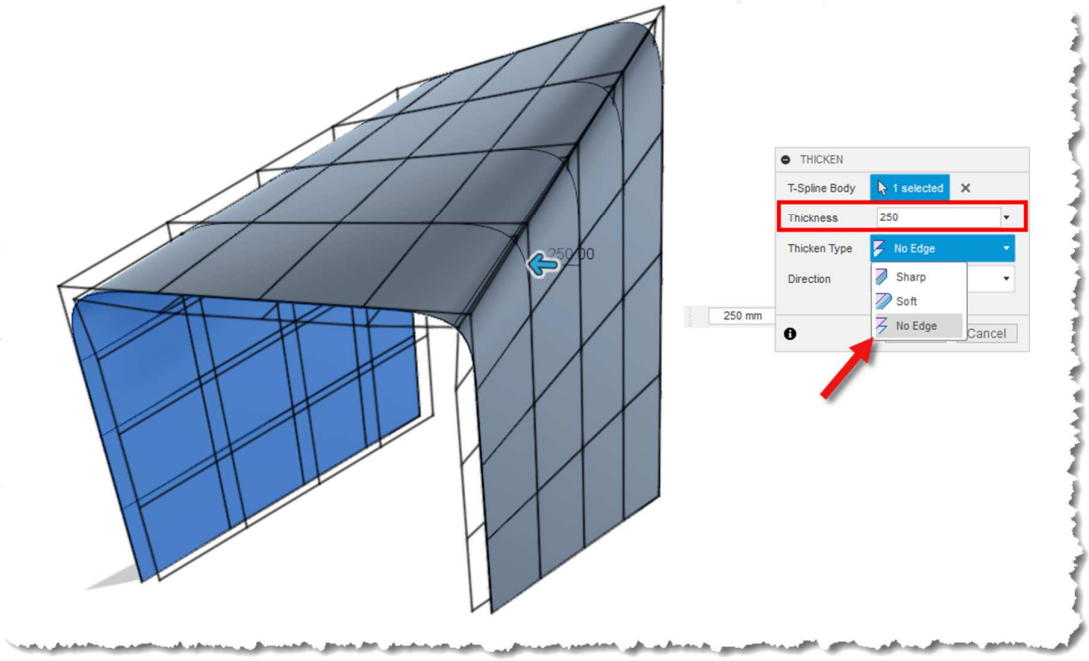
11.2 Internal and External Shells

Fusion 360 does not allow to offset surfaces while in the “Sculpt” environment but there is a command which can be used as a workaround and which produces the same result.

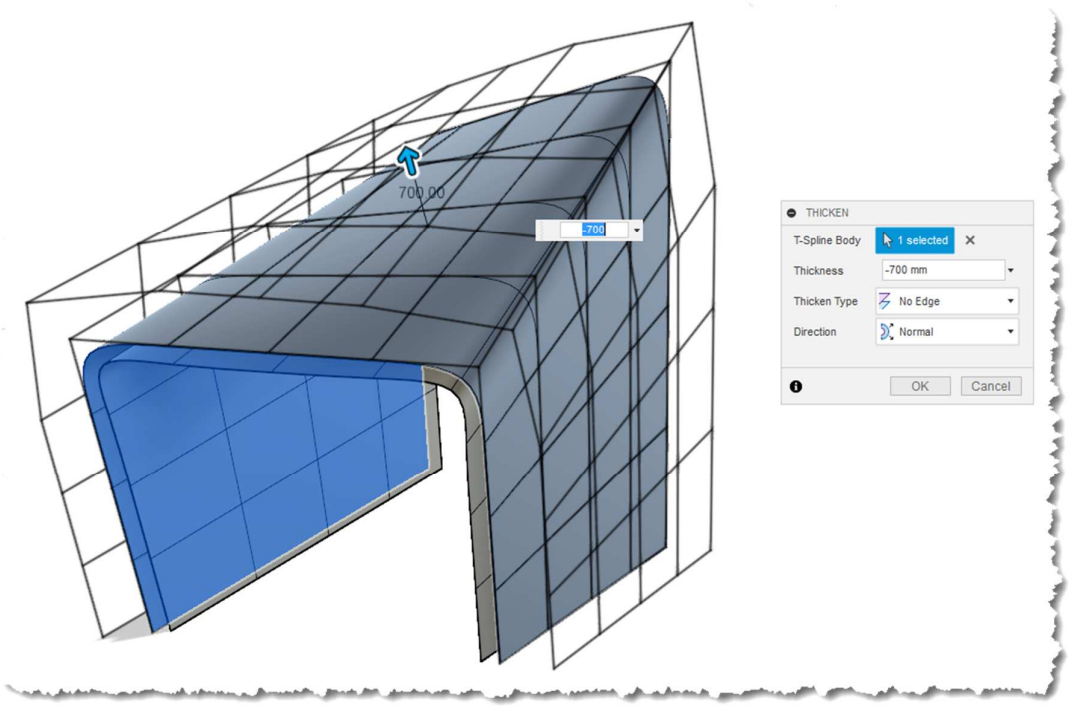
1. Click on the “Thicken” command under the “Modify” menu. This command works in a similar way of the other with the same name that can be found in the Model and in the Patch workspaces;



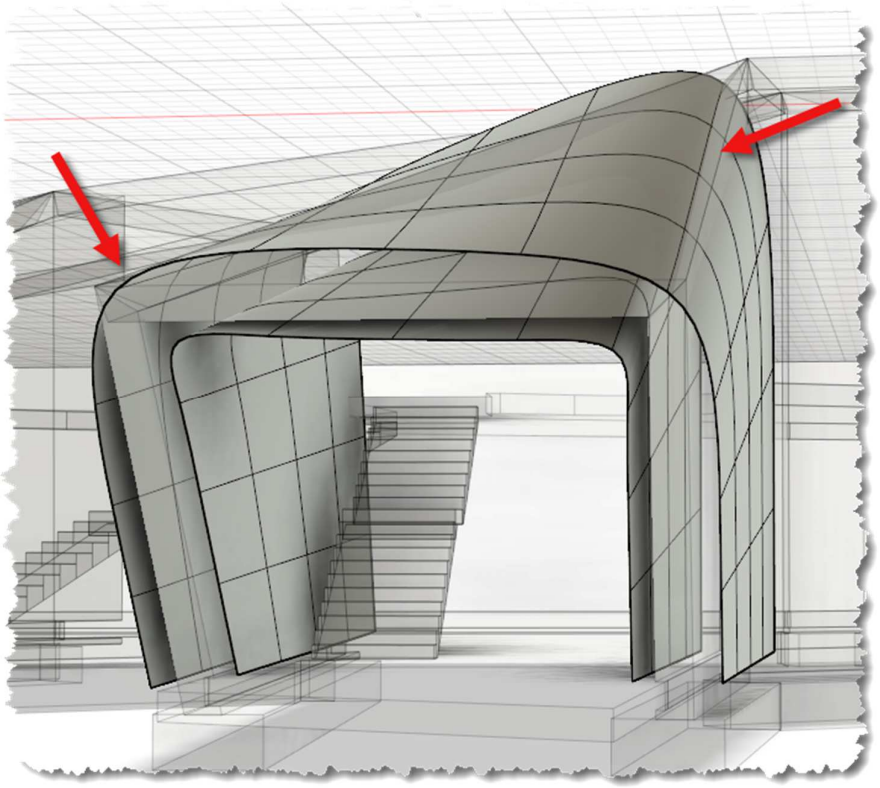
2. Specify 250 mm as the “Thickness” and set “No Edge” as “Thicken Type”. This generates an offset geometry inward;



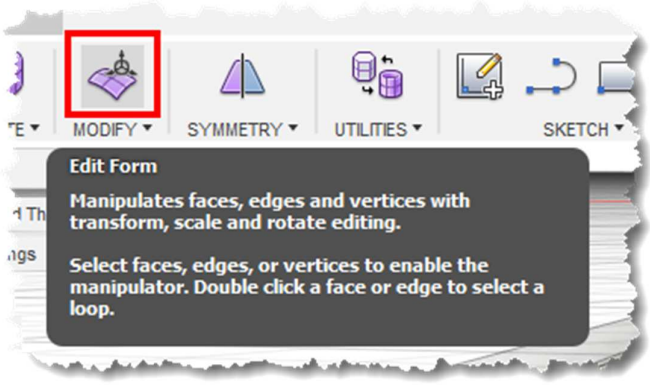
3. Repeat the same outwards, specifying 700 mm as the Thickness;



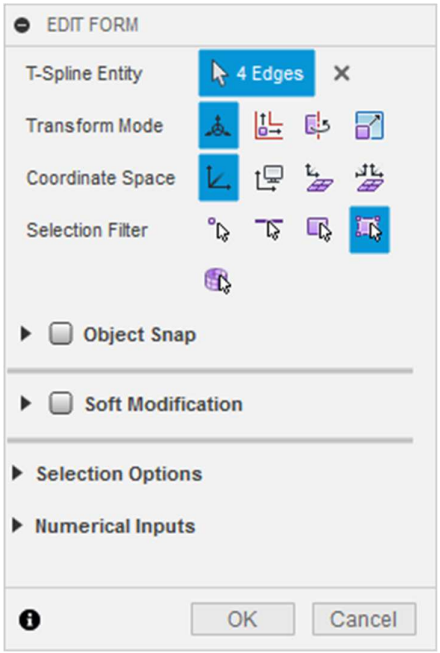
4. Switch off the visibility of the base surface and make the building structure component visible; the external corner curvature is too big so that the surface intersects the concrete structure;



5. Use the “Edit Form” command to be able to manipulate the surface sub-components such as vertices, edges and faces;

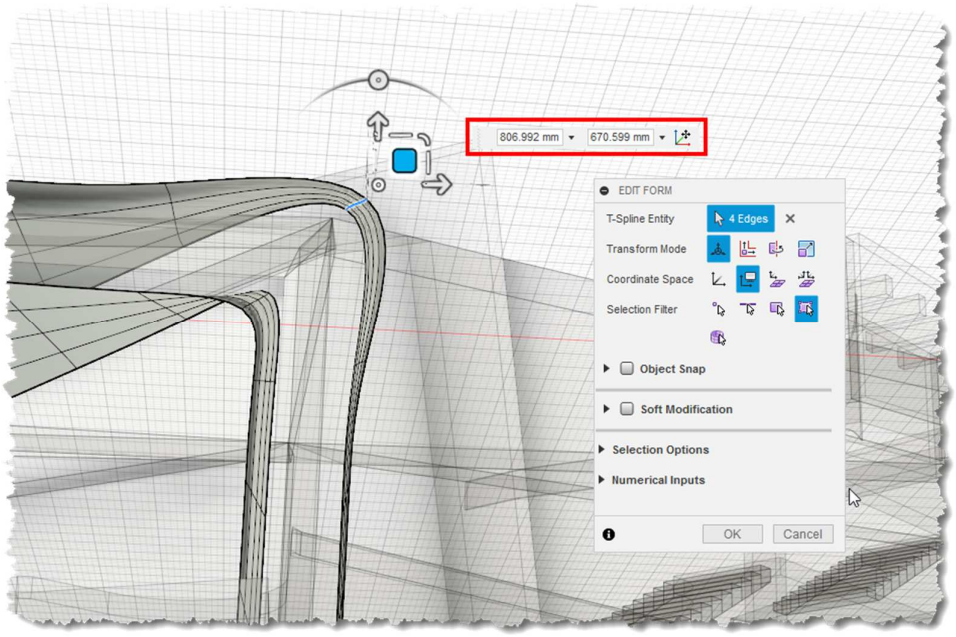


6. The “Edit Form” control dialog allows to set several options in order to increase the efficiency while manipulating the shape;
    a. The “Transform Mode” allows to choose to translate, rotate, scale or to have a combined approach allowing to perform the three kinds of edits (this is the first icon);
    b. The “Coordinate Space” allows to edit elements using the global coordinates, the plane perpendicular to the point of view (View Space) or local coordinates;
    c. “Selection Filter” allows to limit the editing capabilities to vertices, edges, facets or all of them;
    d. “Soft Modification” is for “expanding” the edit effects to surrounding elements; this propagates the changes depending on a parameter called “Weight”. The main scope of the “Soft Modification” is obtaining smooth transitions inside the surface;

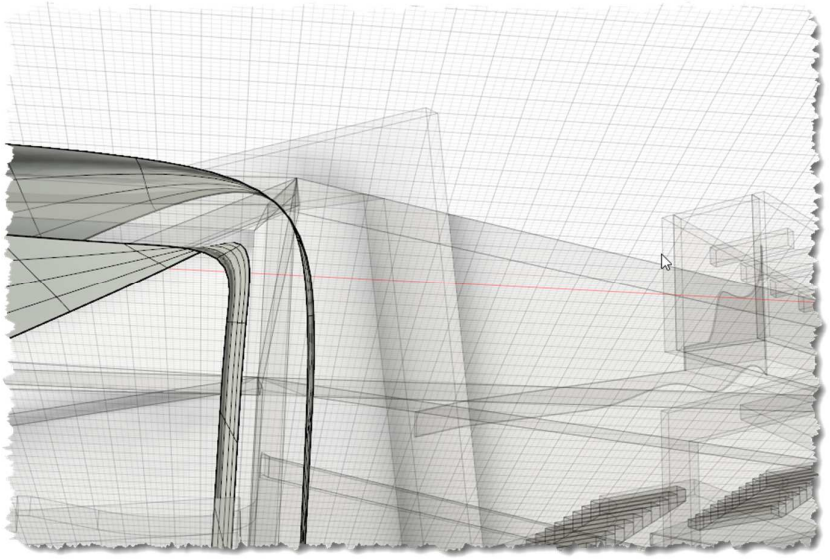


NOTE
Selection Space transforms selected objects as a group in the direction of the axis of the manipulator. **Local Per Entity** transforms each of the selected objects locally, relative to the direct of the axis of the manipulator.

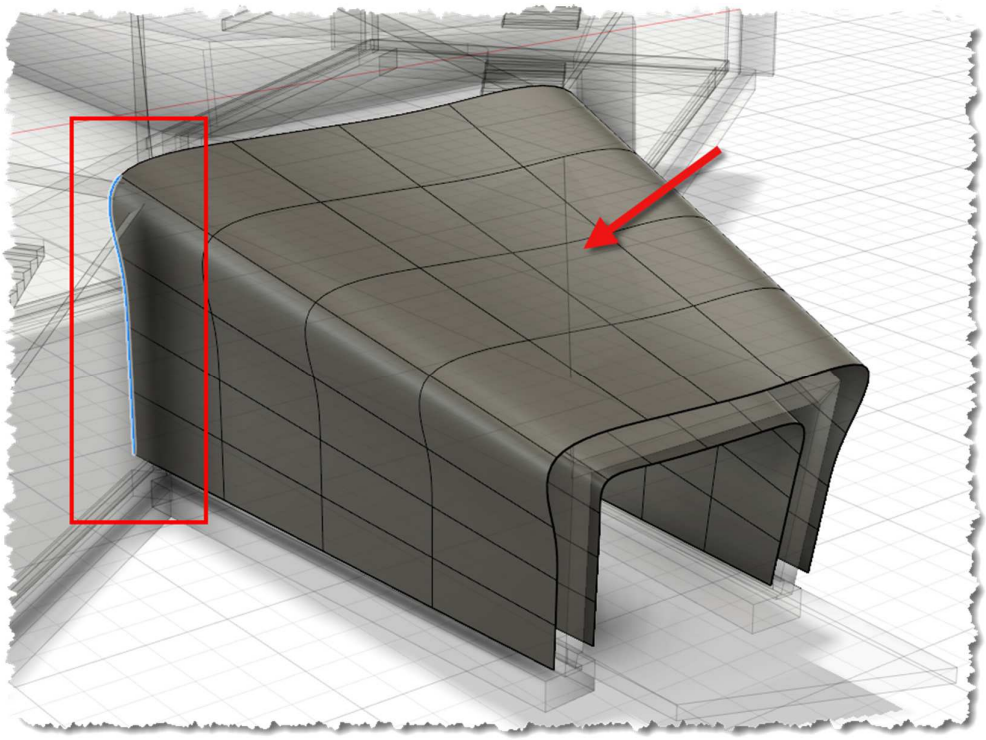
7. Select the “corner” edge and drag it as shown in the next image; use the “View Space” Coordinate Space (second icon); use the vector components in the red box as a guide for the movement of the selected edge;



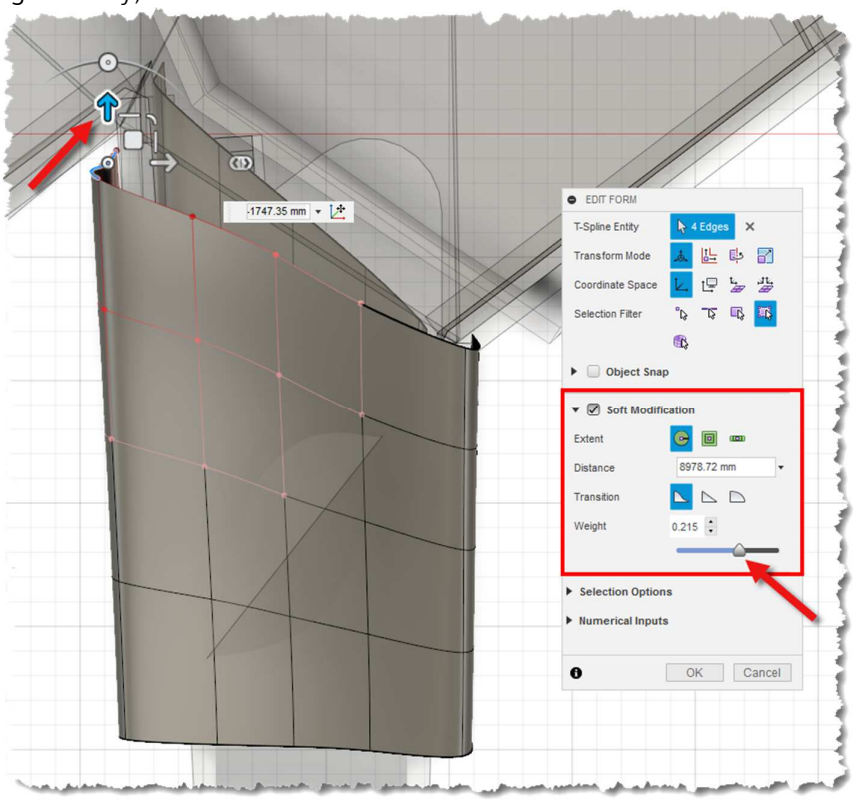
8. The result should appear as shown in the picture below;



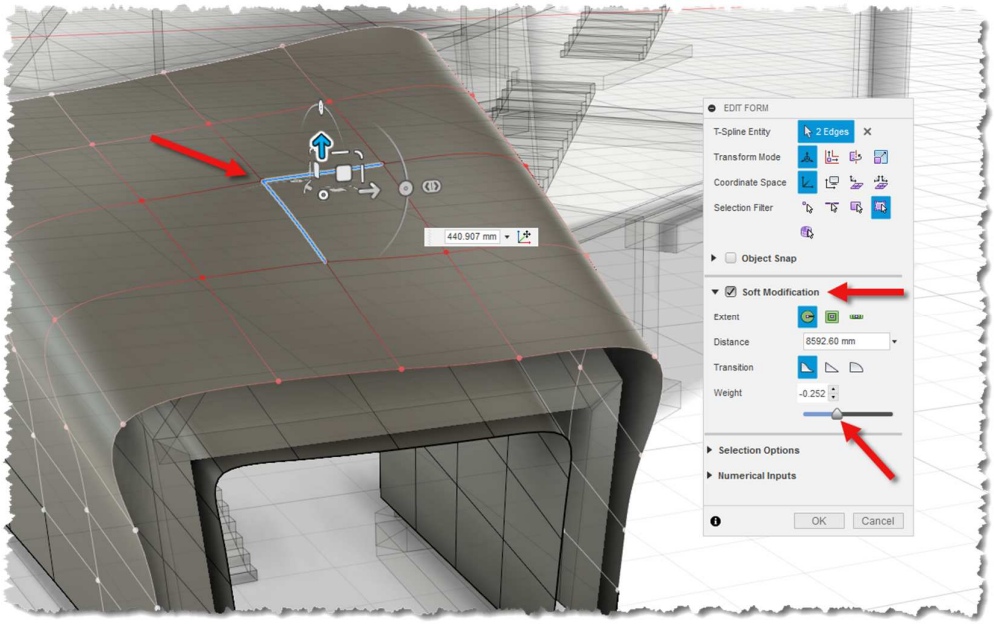
9. Use the same technique to move the highlighted edge away from the structural walls intersection and to raise the top faces to avoid the clash with the roof edge;



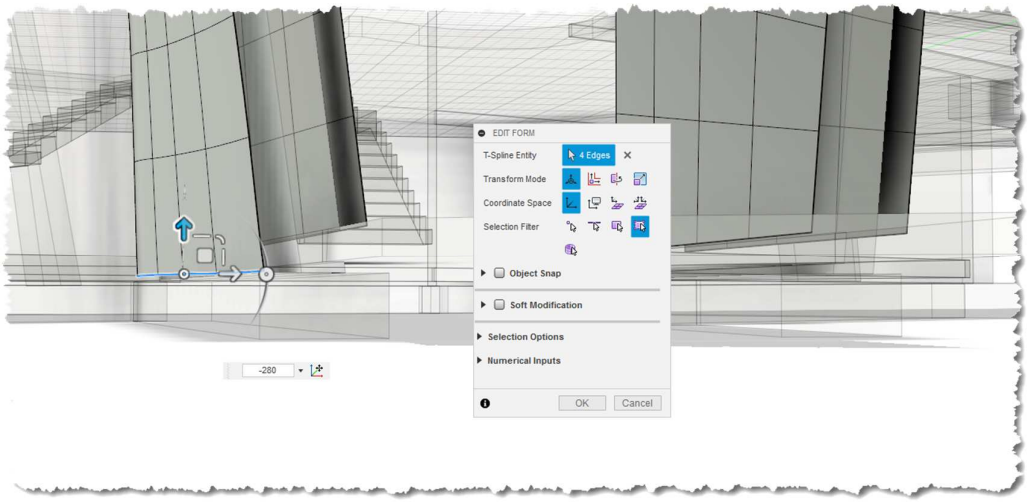
10. In this case, use the “Soft Modification” to avoid generating “twirls” and irregularities on the geometry;



11. Move up the two selected edges to avoid the interference with the roof;



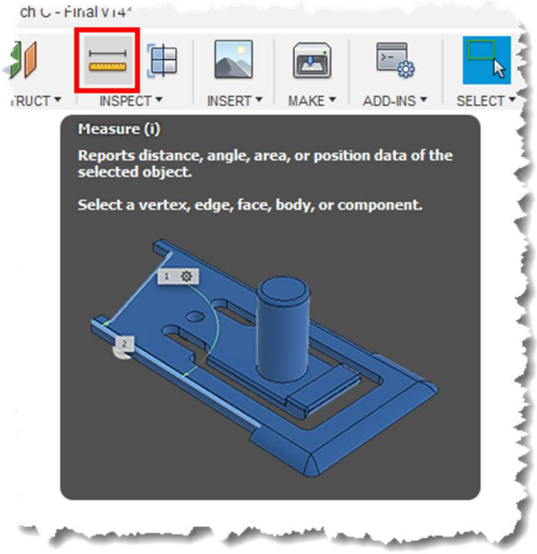
12. Use the same command to move the outer shell lower edge to reach the foundation top face; lower it by 280 mm;

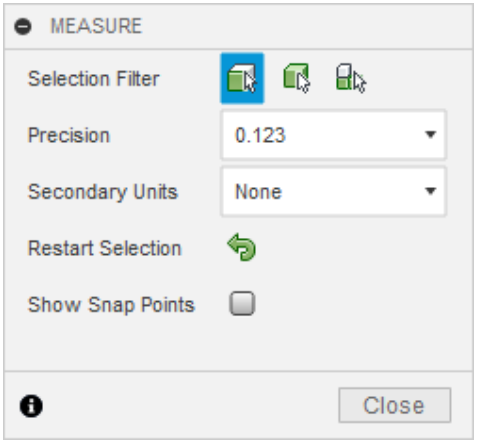


11.3 Parapet Creation

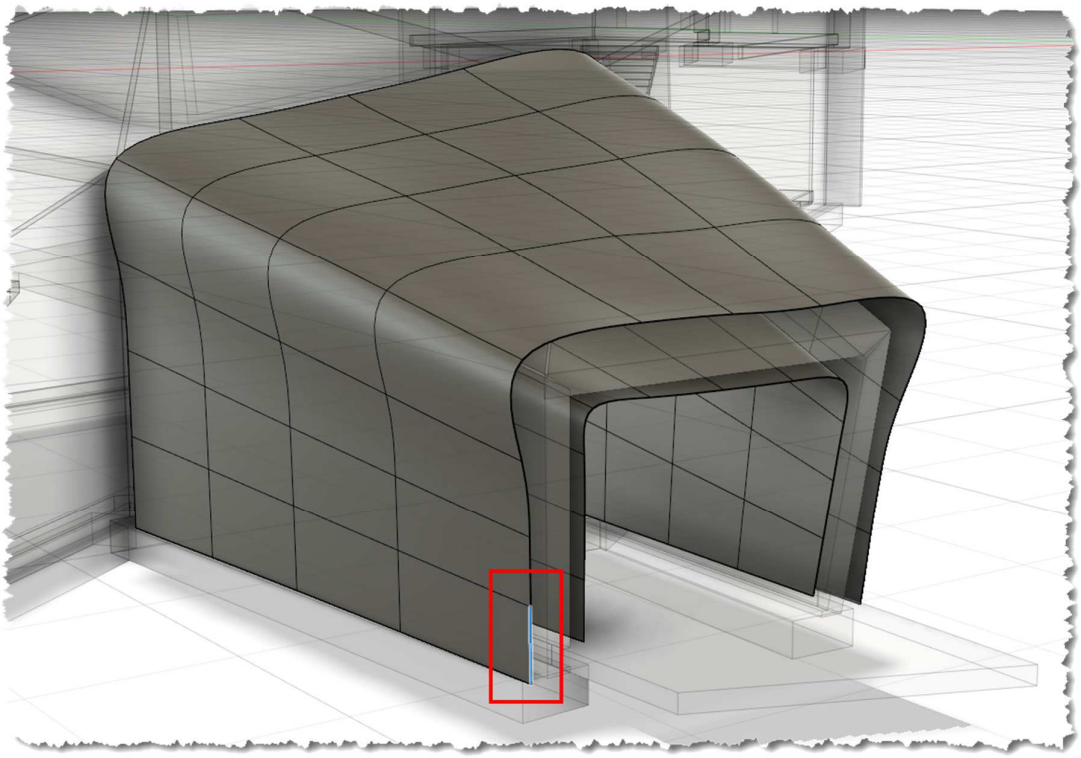
The next step is related to the creation of the parapet and to connect the internal and the external shells.

1. Select the external lower edge only; this will be used to drive the parapet definition. Measure the height of the edge from the floor top face and if needed adjust it to be around 800-1000 mm: for this task use the “Measure” command (type “I”);

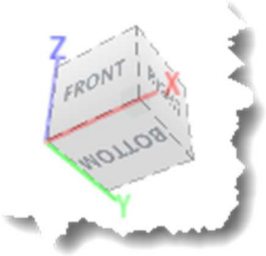




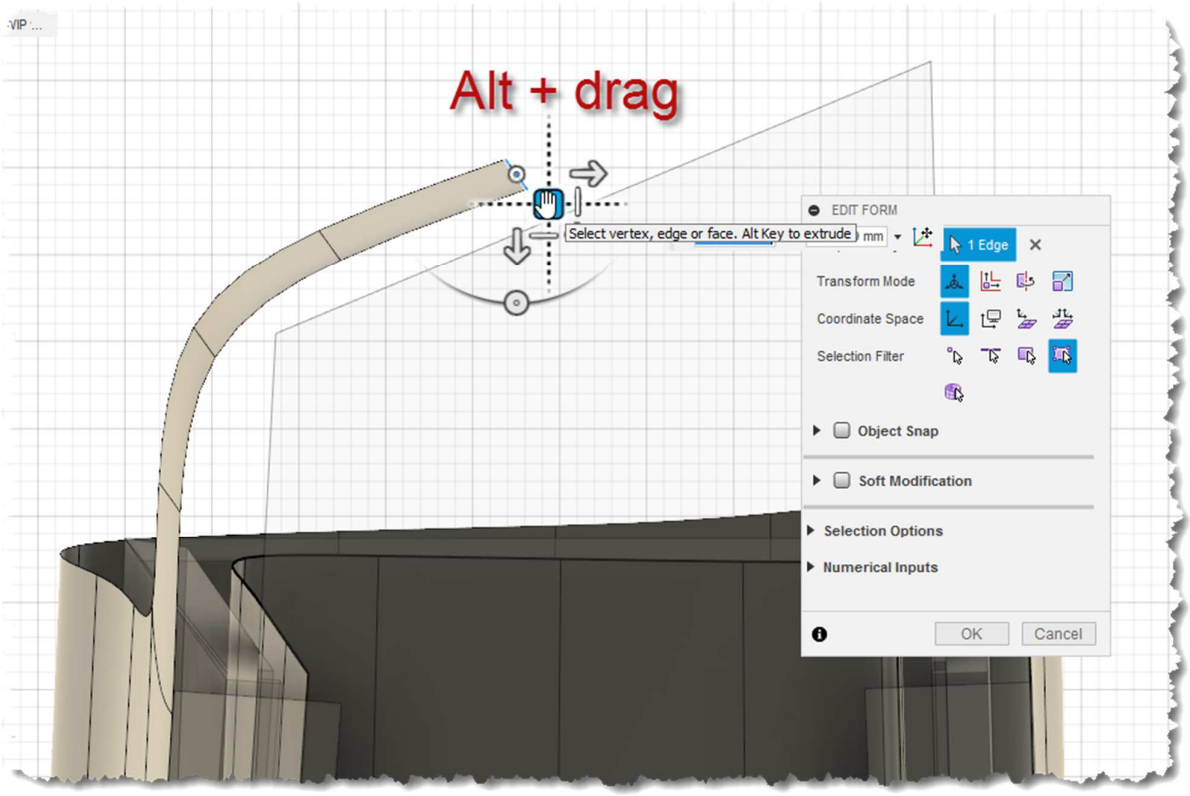
13. Next image shows the edge to be selected;



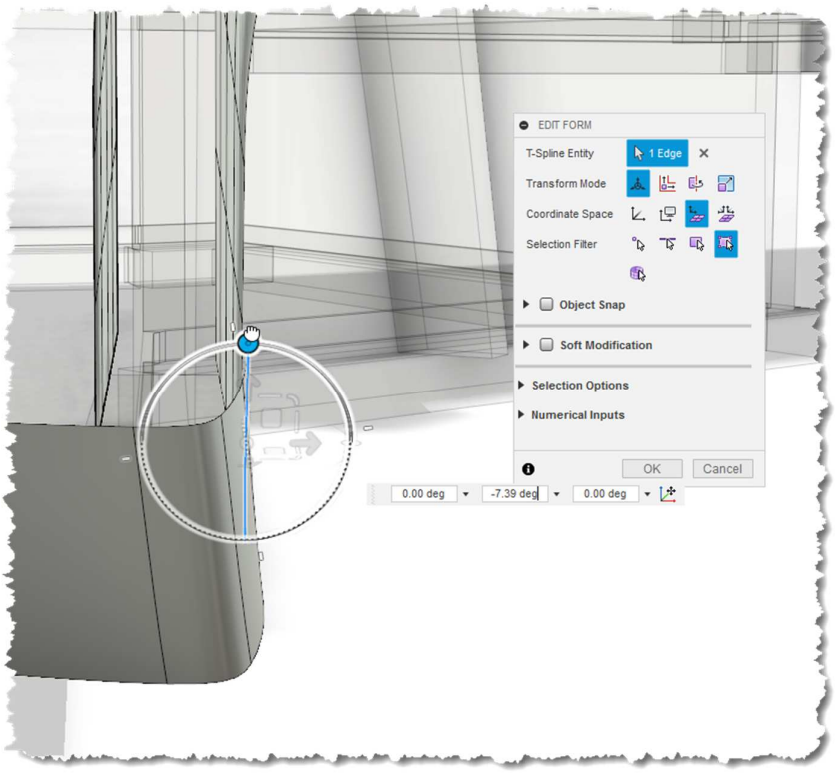
14. Click on the bottom face of the View Cube to move the point of view below the building. In this case this allows to have a better point of view of the slab so that it would be easier to model the parapet boundary;



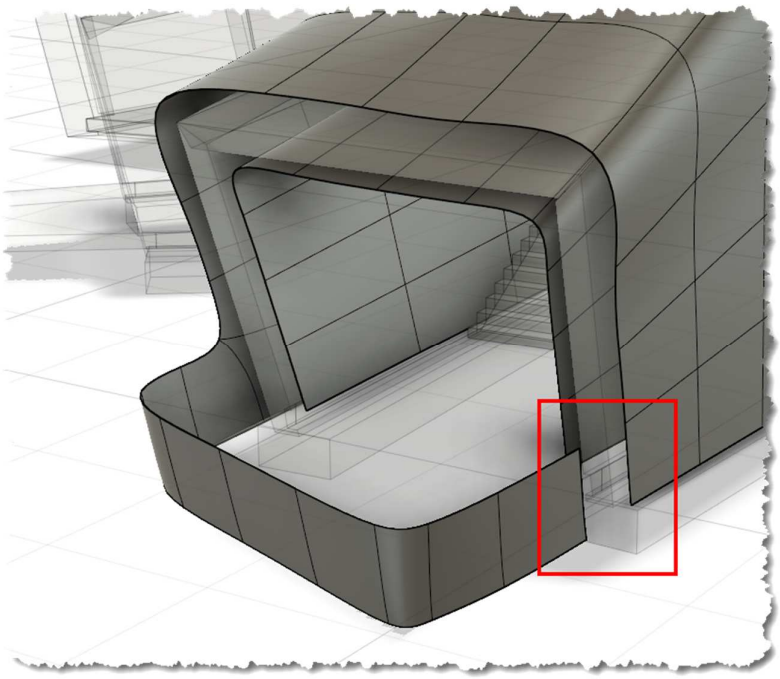
15. Use the “Edit Form” to start the parapet definition; keep the **Alt** key pressed while dragging the edge to create new faces. Follow the floor perimeter and stop the modeling about at 500-1000 mm of distance from the other external shell edge;



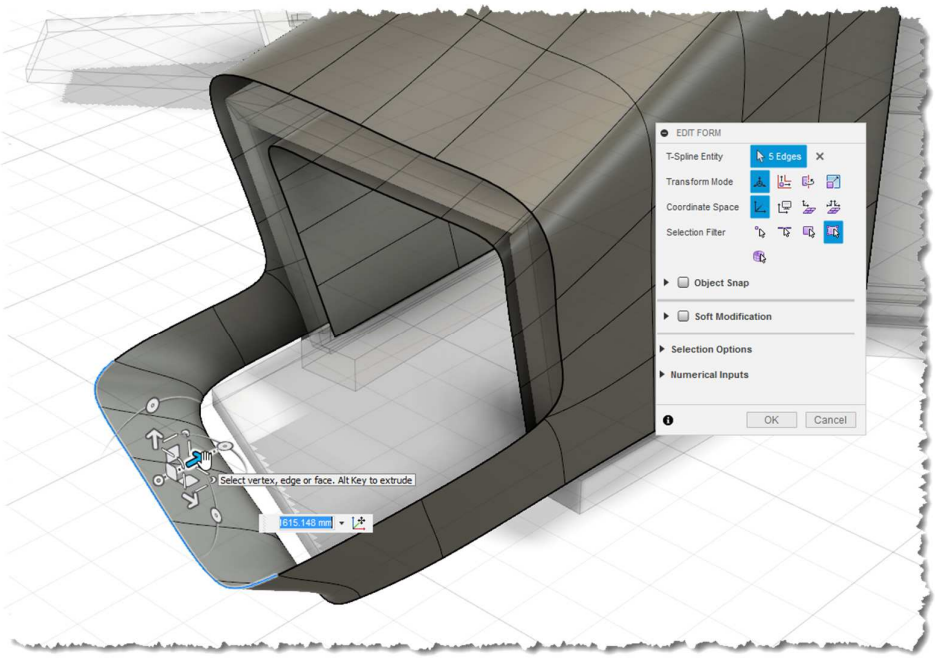
16. If needed, change the rotation of the last elements in order to match the slope and orientation of the related element of the external shell with whom it will be connected to finalize the parapet modeling task;



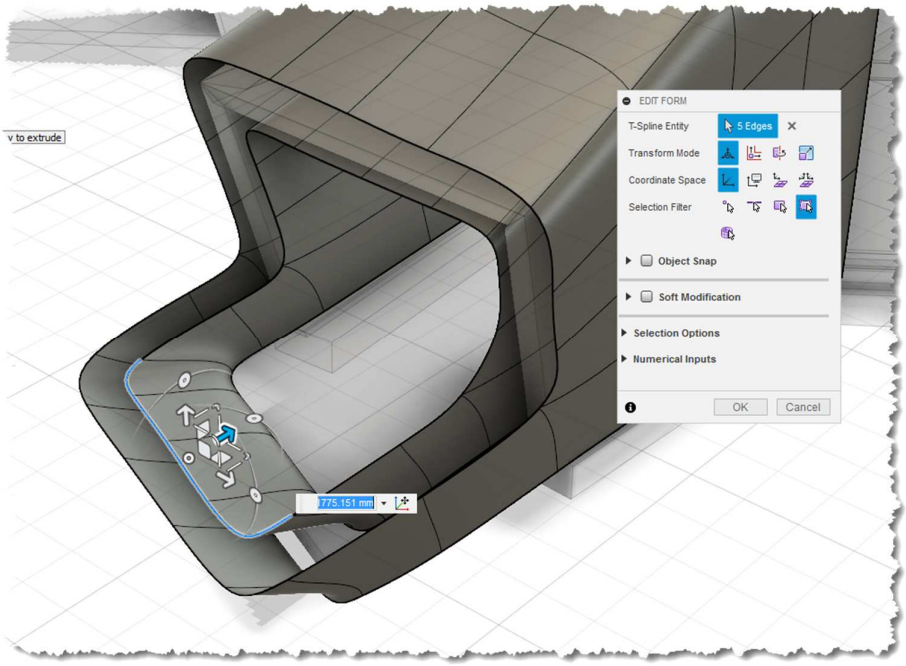
17. Use the “Merge Edge” command to join the two edges;



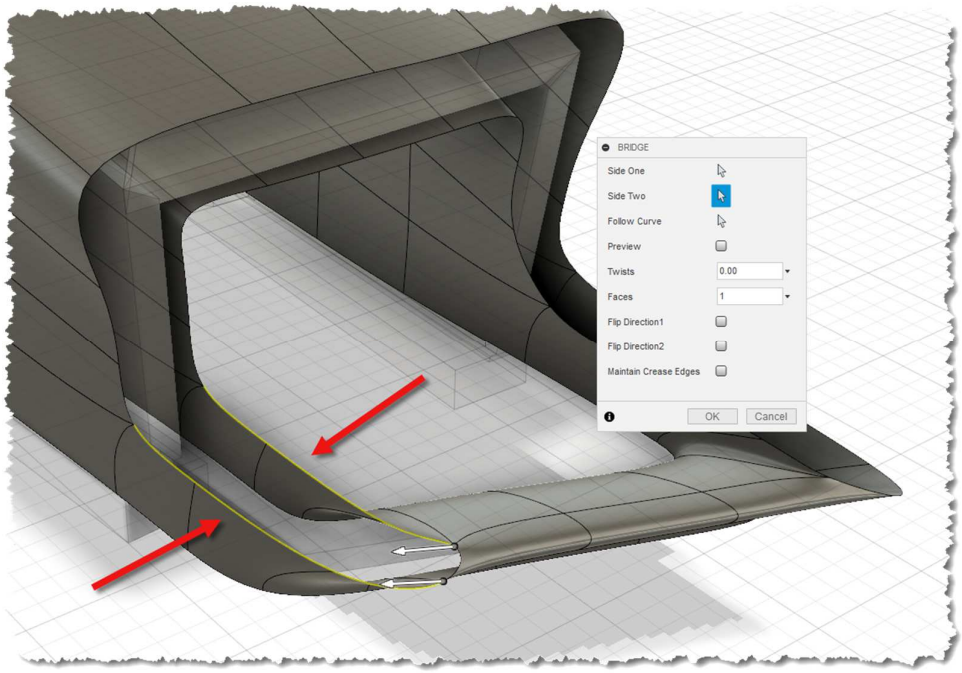
18. Use again the “Edit Form” feature to drag the top edges (whose amount depends on how many segments have been added using the **Alt + drag** feature) of the parapet outwards;



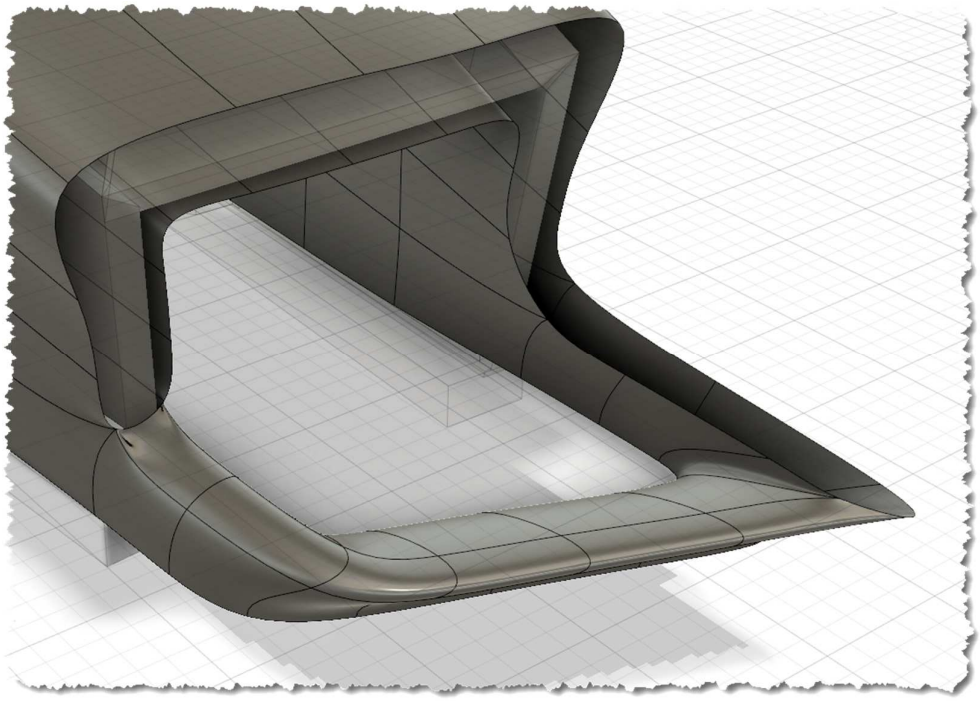
19. Repeat the same procedure to create the internal face of the parapet and drag the upper edges outward to have them closer to the related edges of the outer face;



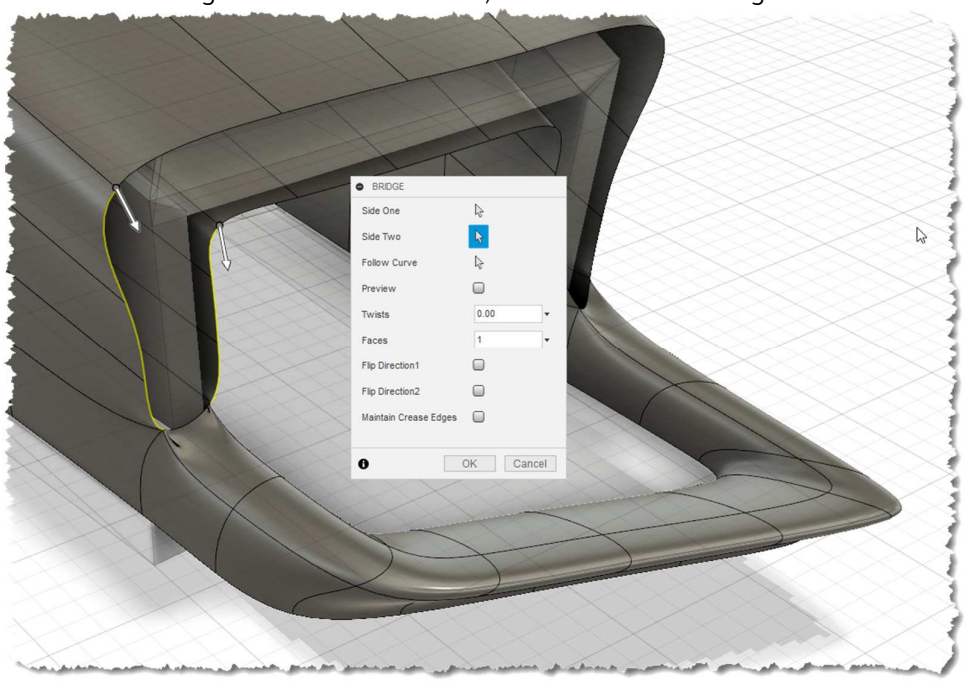
20. Use the “Bridge” command to start joining the external and the external faces; set “1” as the number of faces to be added with the bridge command;
21. Avoid selecting the whole set of external and internal edges; it advisable to proceed step by step instead to maintain a higher level of control on the final shape and to correct possible errors;



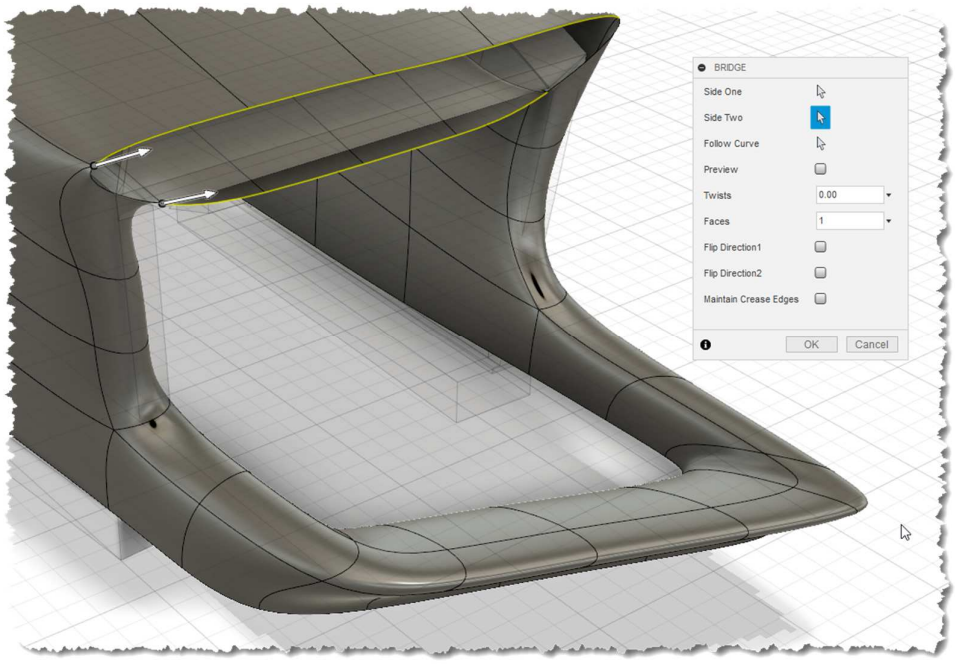
22. Notice that if a single edge remains between the two sides of a “bridge”, this will be merged on the result geometry;



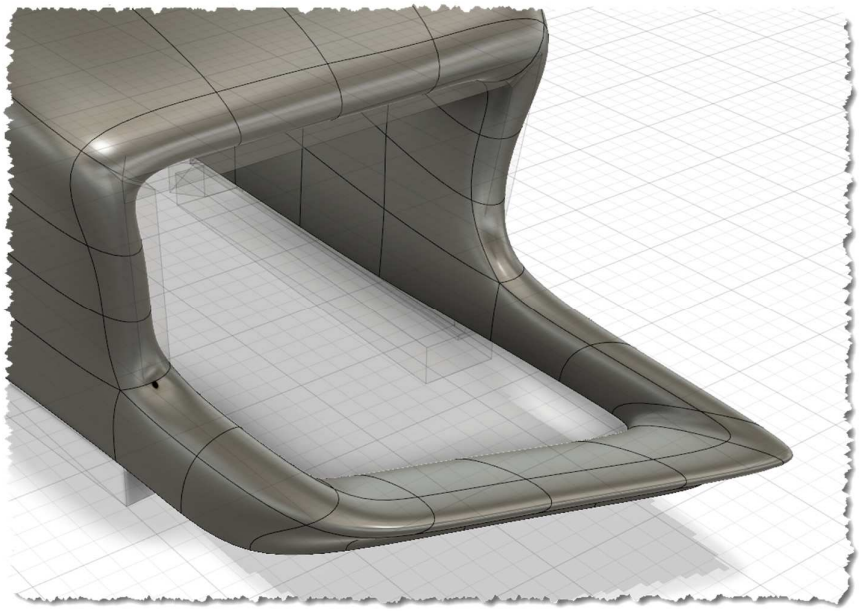
23. Repeat the same procedure for the vertical sides of the opening; be sure to select the same amount of edges for both the sides, otherwise the “Bridge” command may fail;



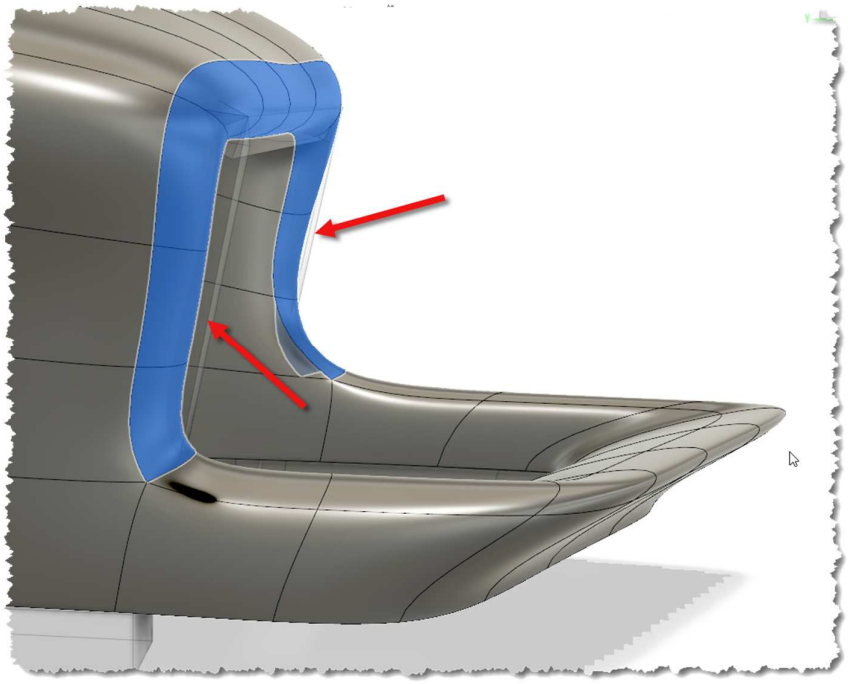
24. Repeat the same on the other vertical side;
25. Use the same command to close the hole on the top side;



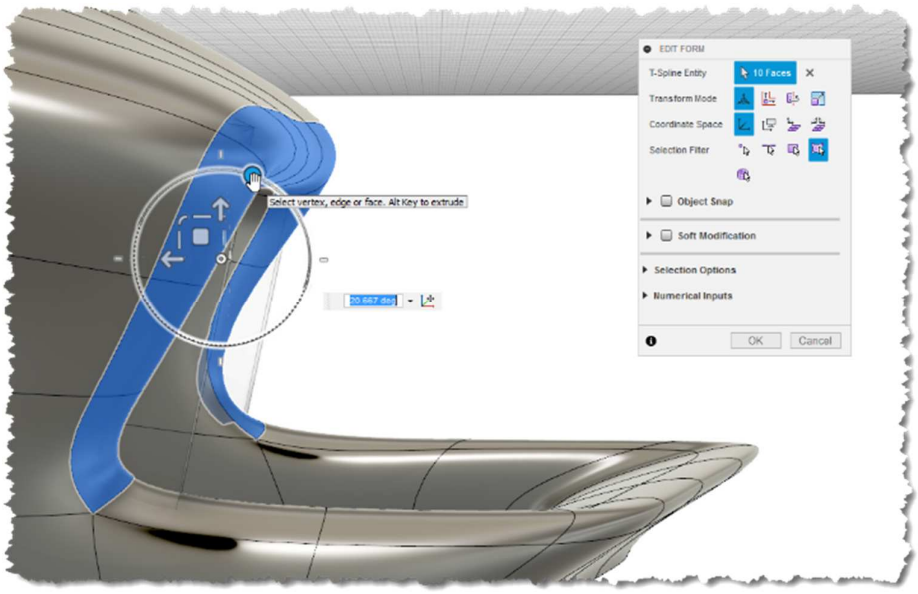
26. The result should appear as that one shown below;

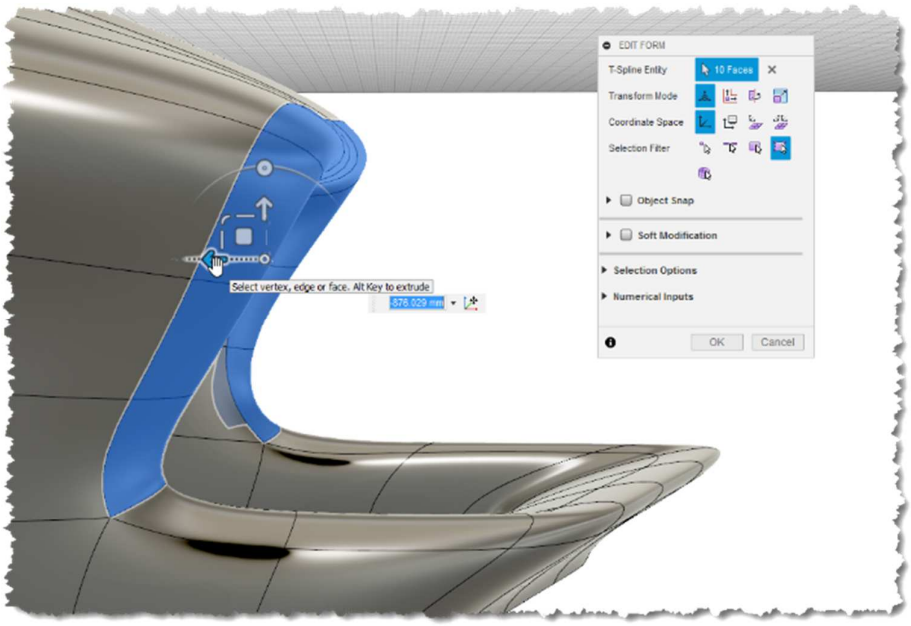


27. Notice that there are some interferences between the cladding shell and the concrete structure. These are clearly visible on the next picture;

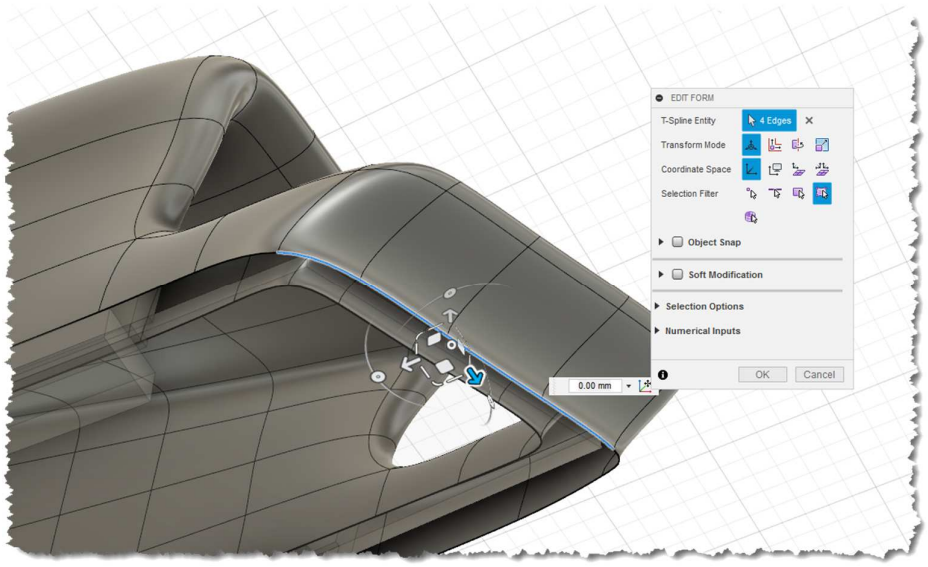


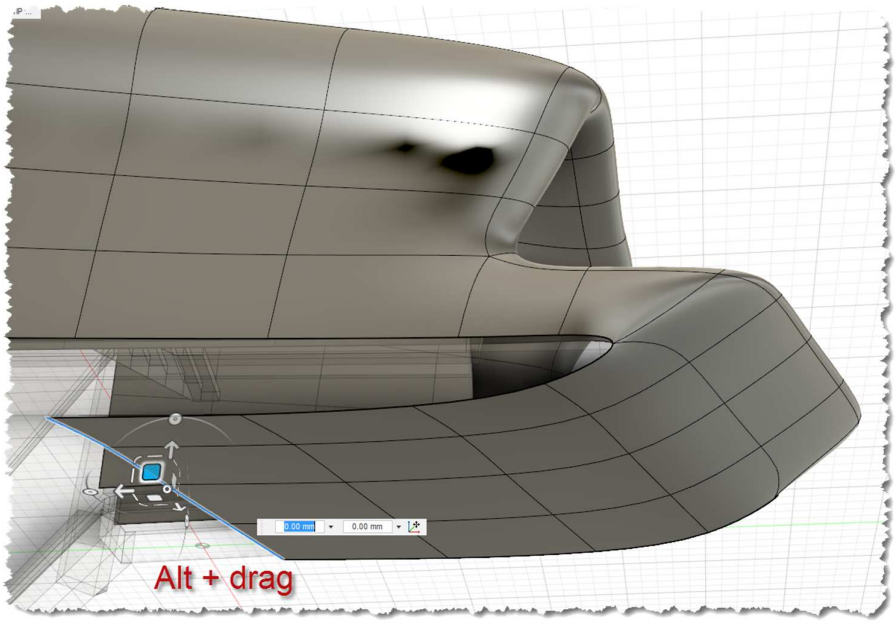
28. Select the facets indicated in the previous picture and use the “Edit Form” command to rotate and to translate them outwards;



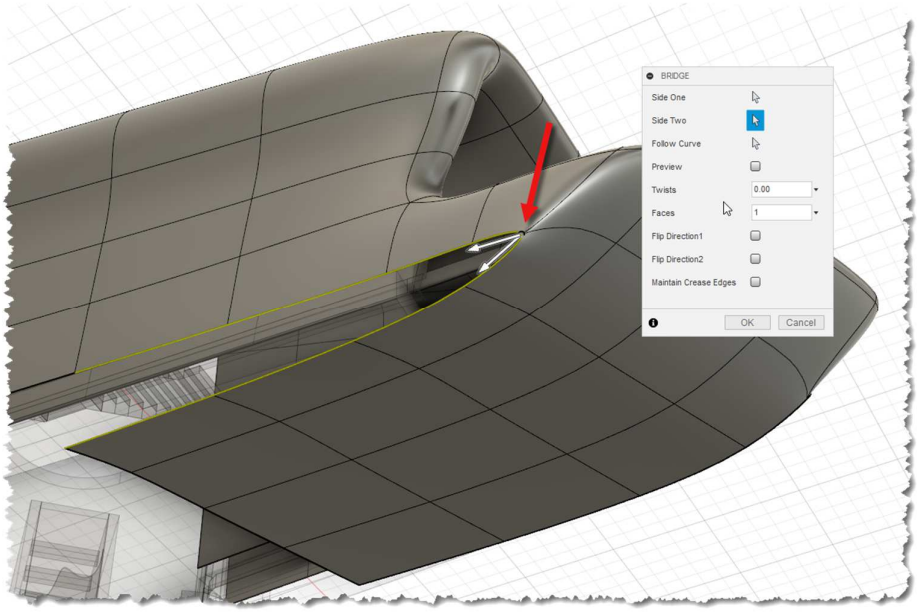


29. Use the “Edit Form” again for creating the bottom shell. Select the bottom outermost edges of the parapet and drag them while pressing the Alt key. Use the most appropriate “Coordinate Space”; this can vary depending on the surface shape and orientation and on the specific modeling preferences;

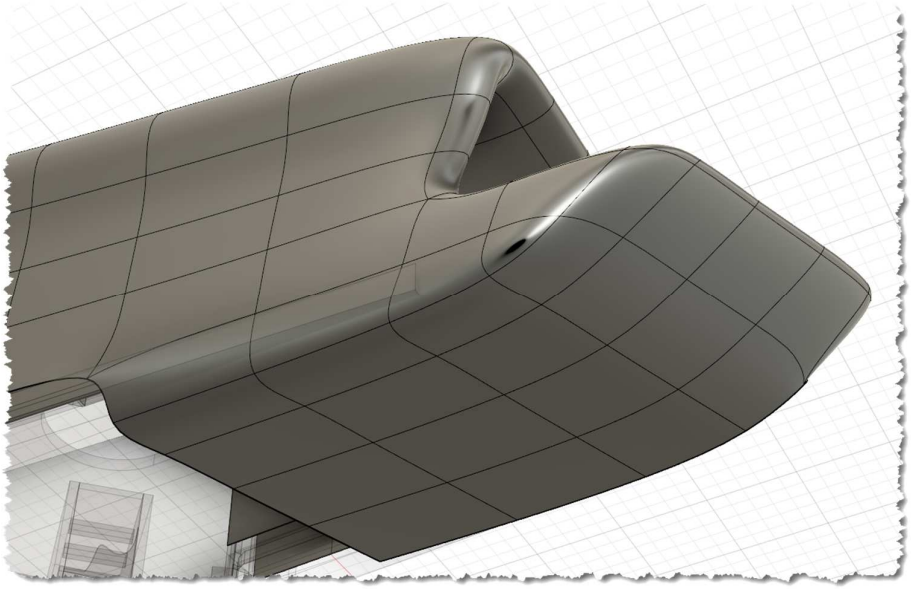




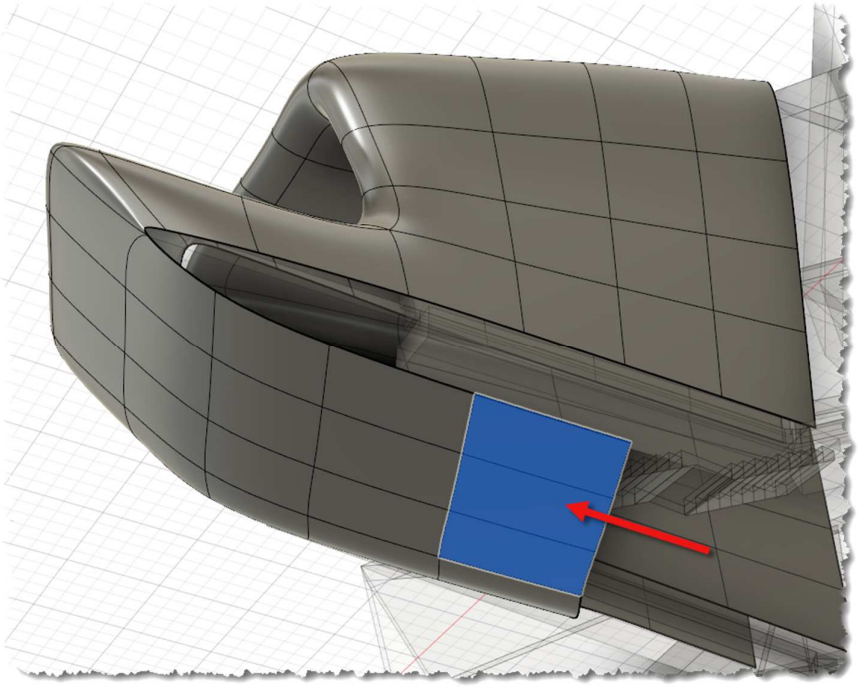
30. Create a Bridge to join the side edges. Notice that in this case the two sides share a vertex;



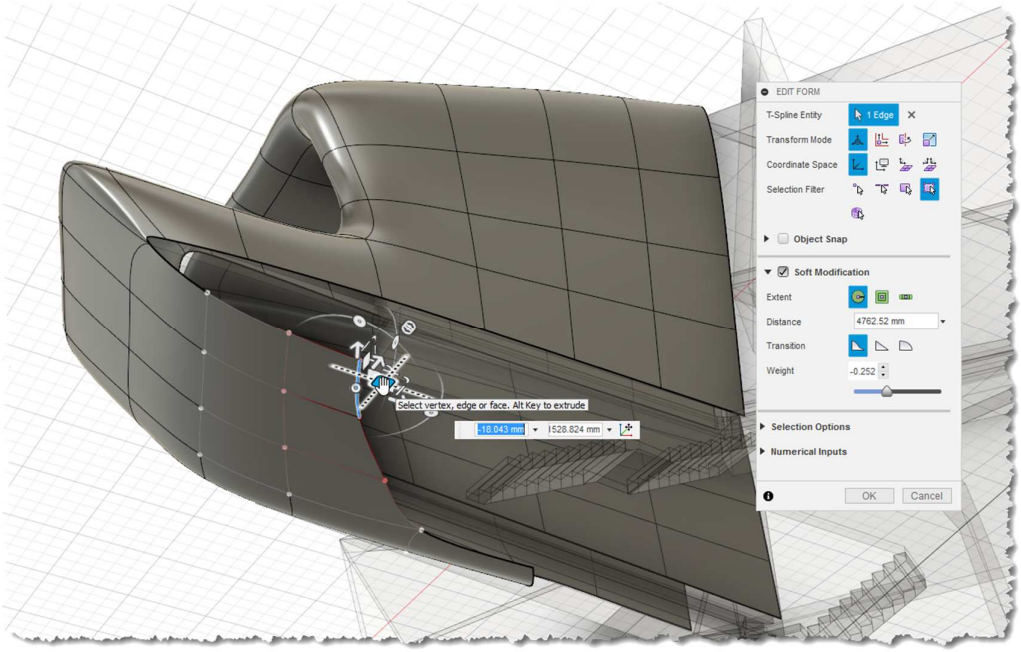
31. The result is a continuous and smooth surface;



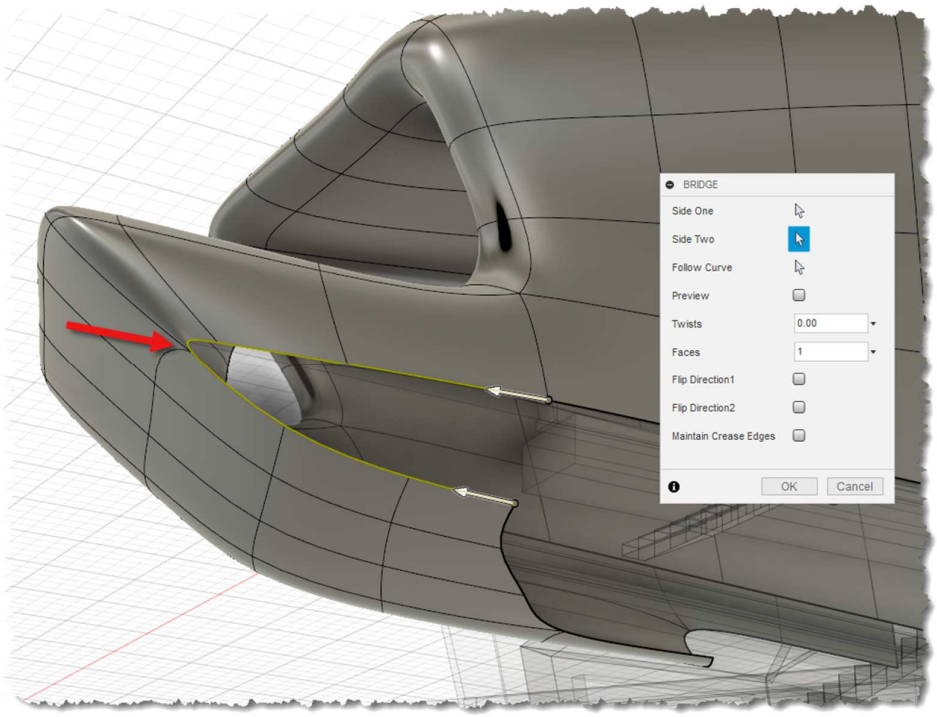
32. The right side of the building branch has a footing which extends more than the other one; this means that the cladding shell must be edited in order to avoid the interference with that structural member. Select and remove the highlighted facets;

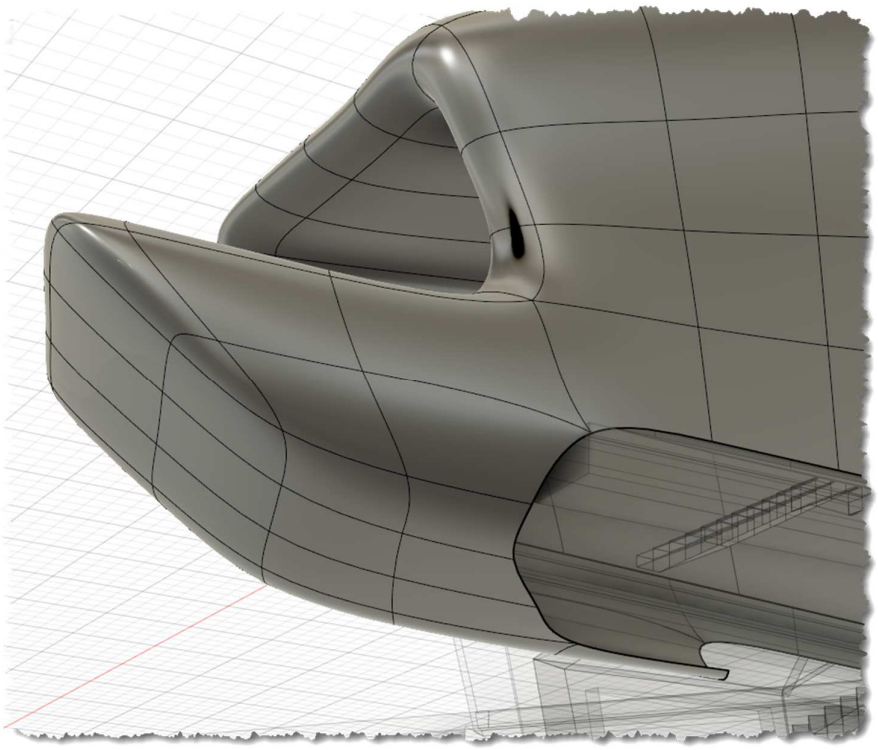


33. Edit the vertices using the “Soft Modification” option to adapt the shape to the footing geometry;

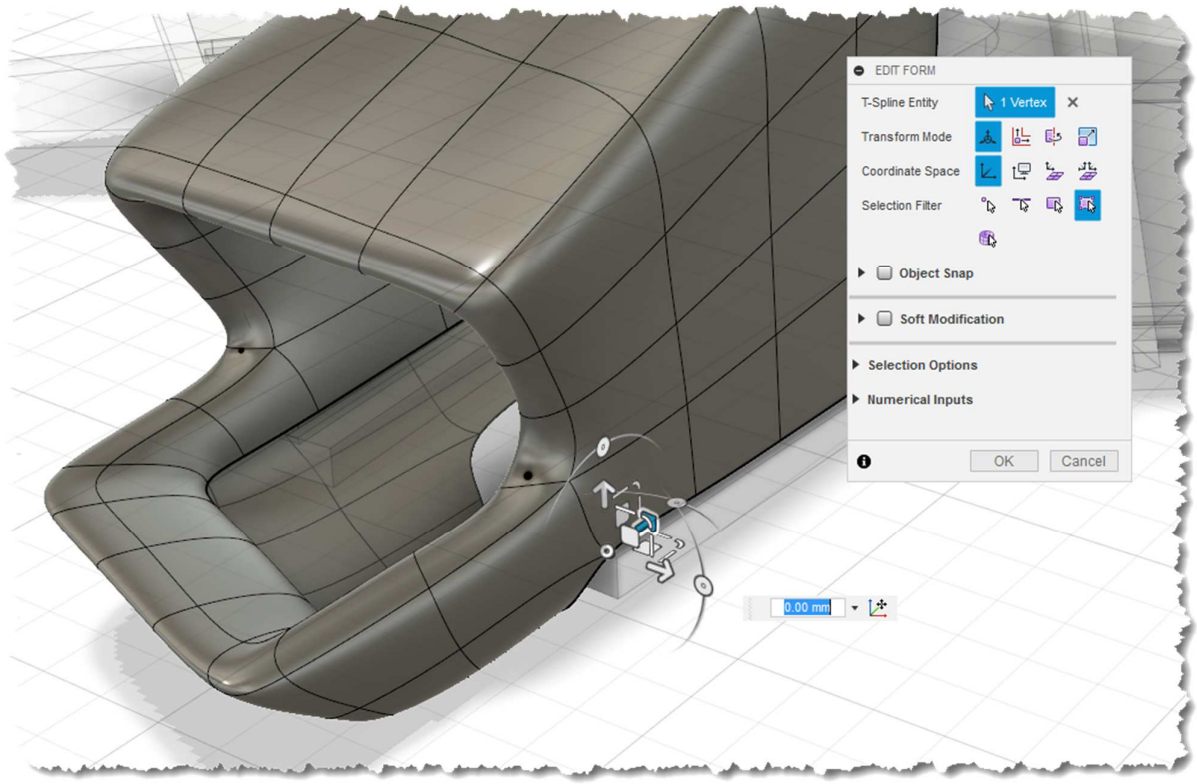


34. Finalize the task by creating a bridge geometry to close the gap. Even in this case the two bridge sides share a vertex;



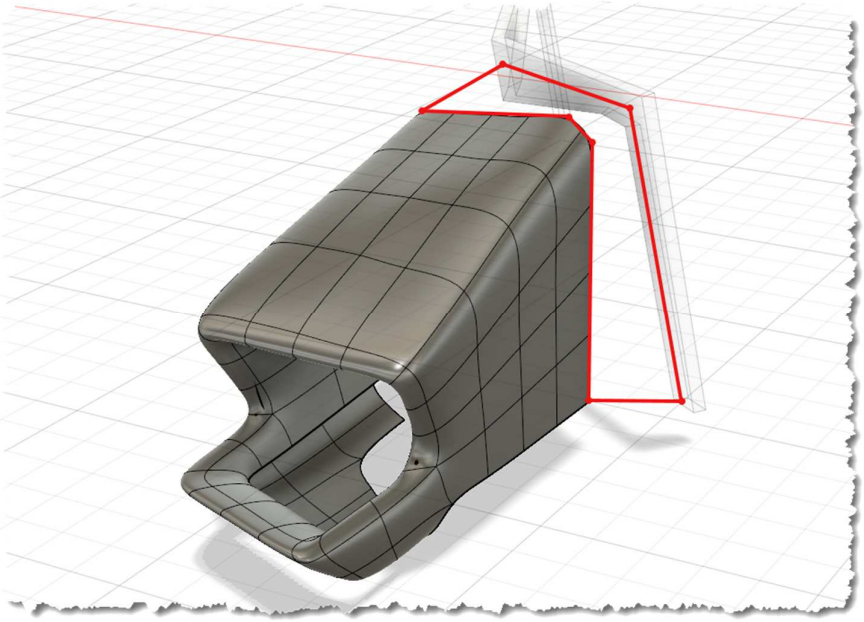


35. Edit the vertices position to refine the spatial relationship between the shell and the structural foundation;
36. Click “Ok” when done;

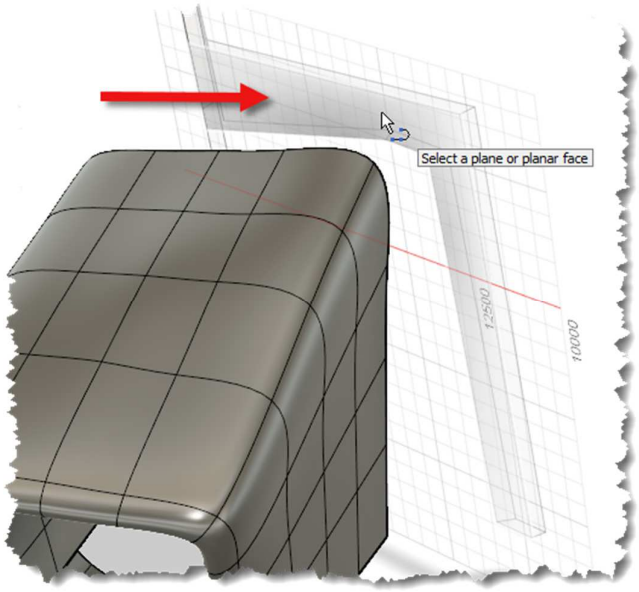


11.4 Connection with the retaining wall

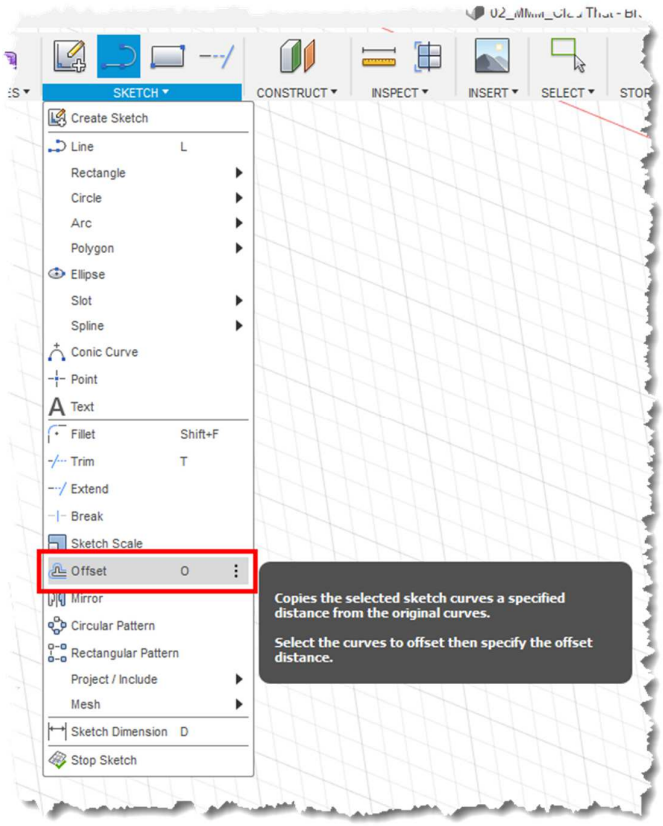
Once the shell geometry has been defined, this must be connected with the retaining wall. The area identified by the red boundary should be filled with a surface connected with the cladding shell.
To obtain this objective, a sketch must be created to drive the creation of a loft surface. After that, the loft edge will be merged with the existing shell edge.



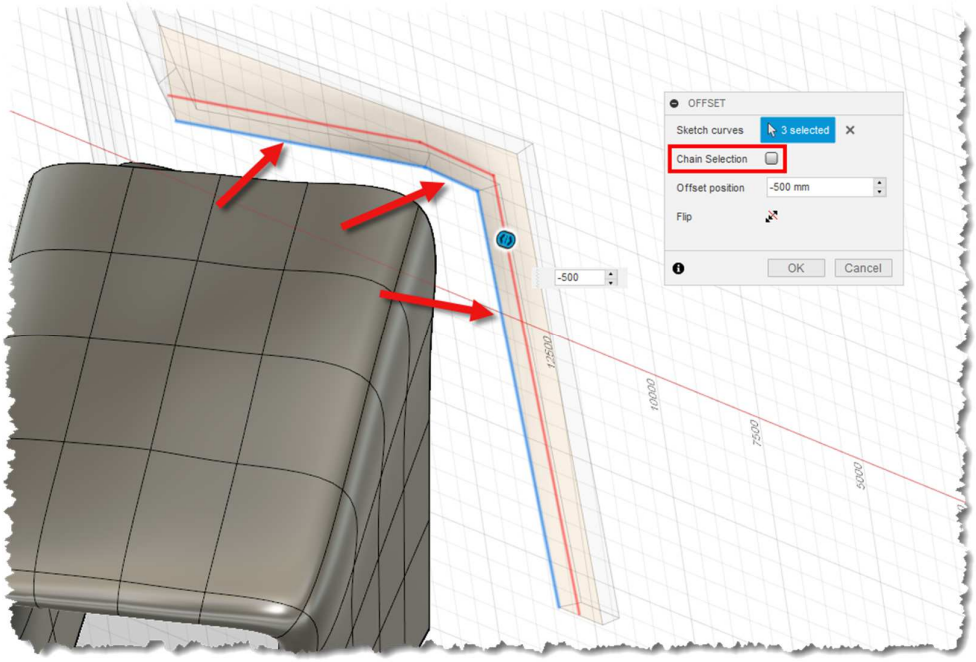
1. Start a new sketch using the retaining wall face as the host;



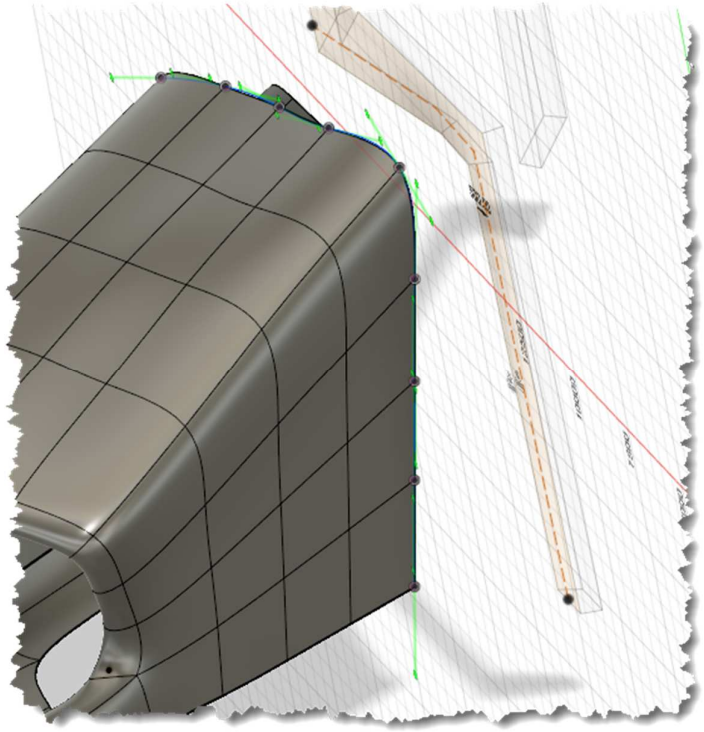
2. Use the “Offset” command to create a 500 mm offset of the lower edges of the retaining wall;



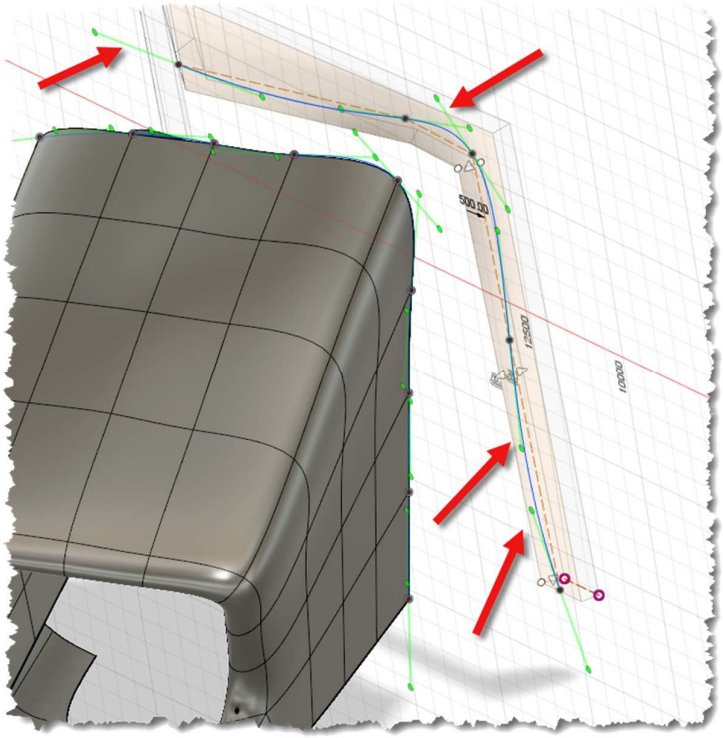
3. Uncheck the “Chain Selection” and select the three lines highlighted below;



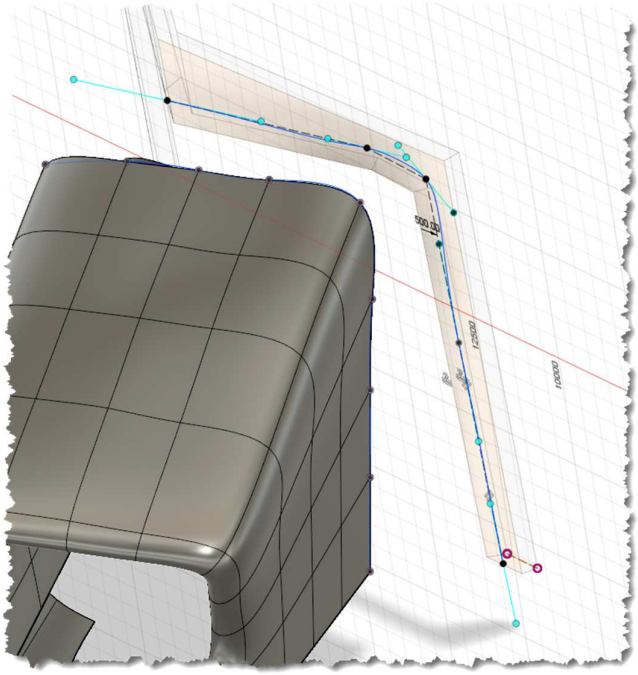
4. Without stopping the sketch definition, create a “Fit Point Spline” and use the shell vertices to drive its shape; also select the offset lines and press “X”: this turns them on construction lines (shown with the dashed line pattern);



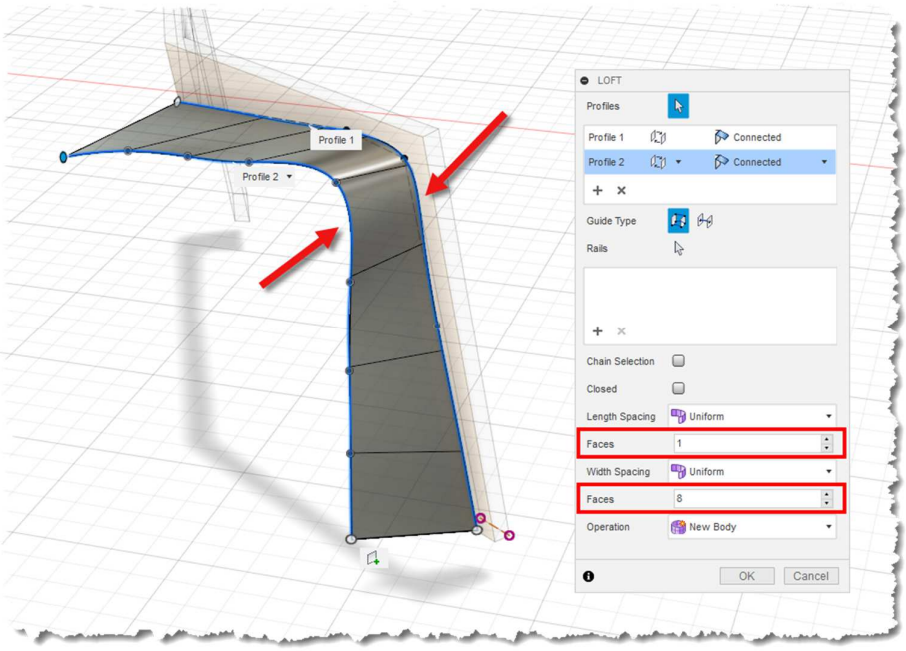
5. Create another spline using the offset edges as a reference; the result shape is not precisely matching with the original linework;



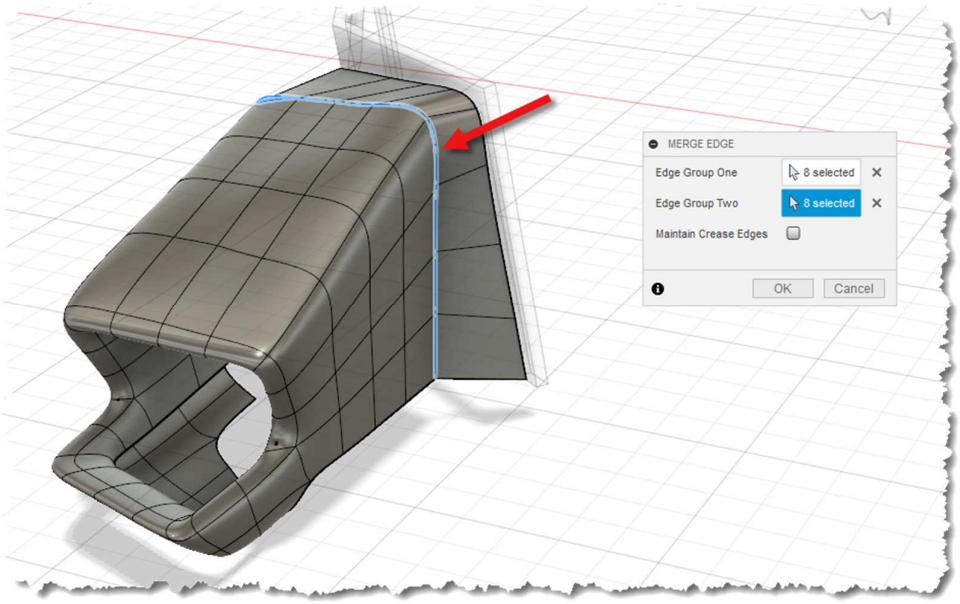
6. Turn off the visibility of 3D bodies and adjust the green handles of spline vertices in order to have the curve the closer as possible to the offset edges. It is not possible to have the two geometries precisely coincident since the offset geometry is a polyline while the spline is a continuous curve;
7. The edited handles change their color and become cyan;
8. Stop the sketch creation;



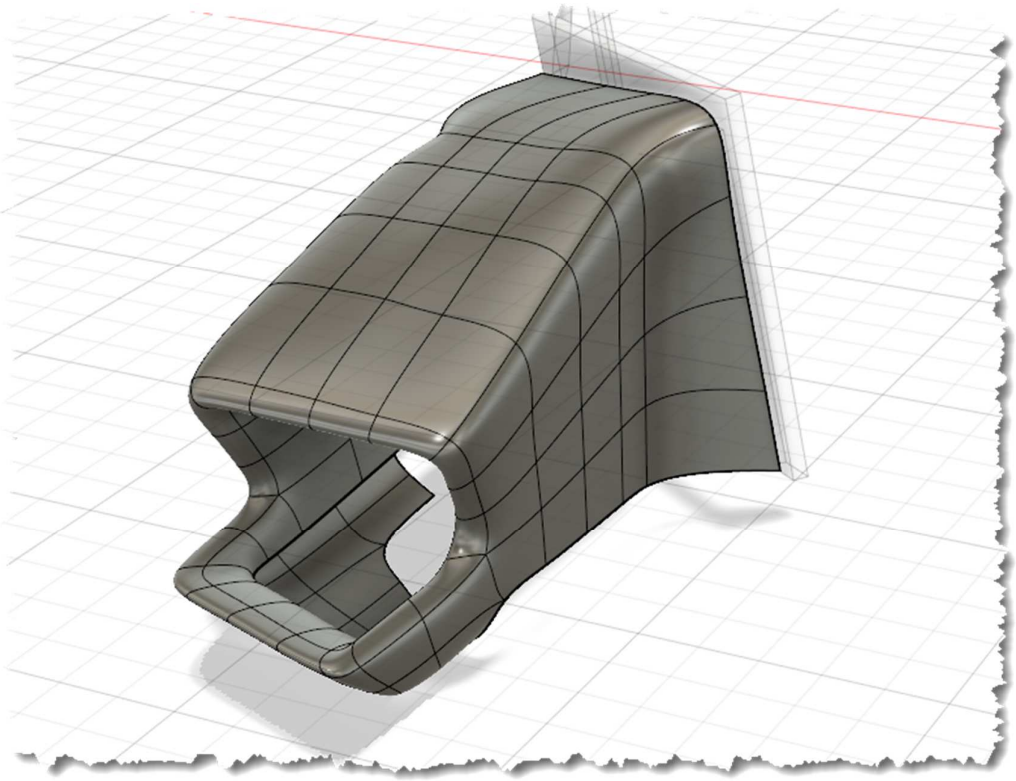
9. As done in previous exercises, use the "Loft" command to create a surface interpolating the two curves; specify the number of divisions as in the below image (Length = 8, Width =1);



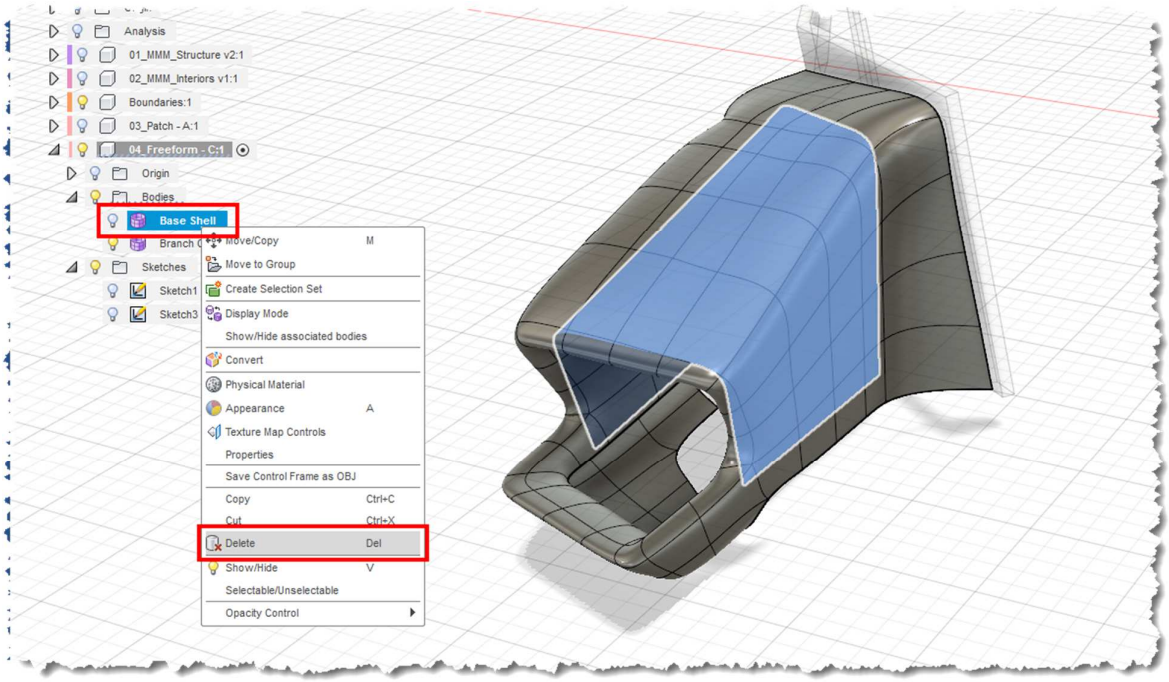
10. Use the “Merge Edge” command to join the two surfaces, obtaining a continuous and smooth shell;



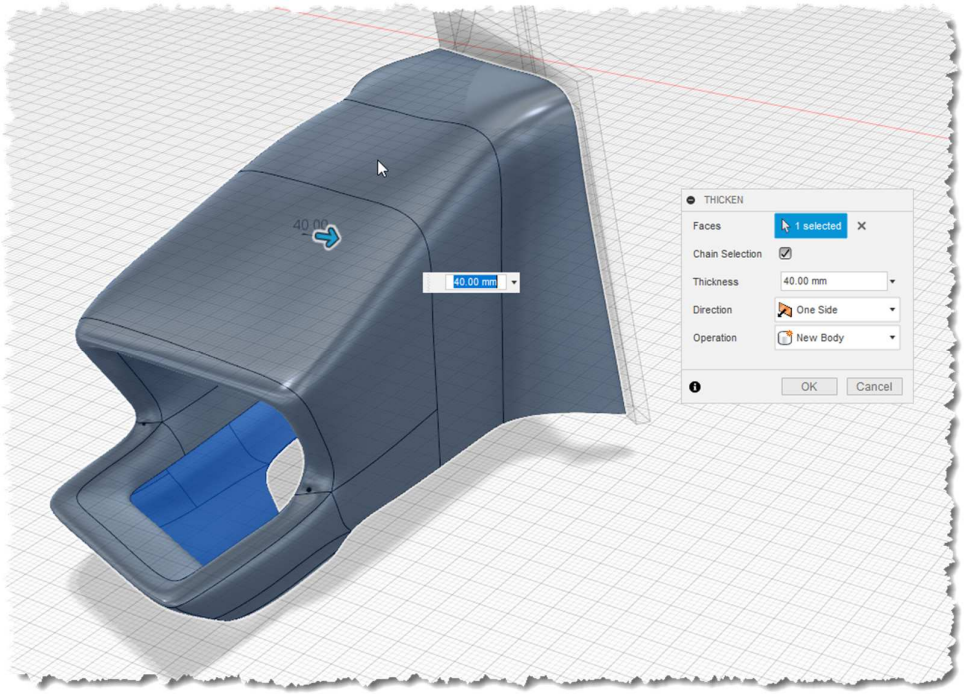
11. The result should appear as in the next picture;

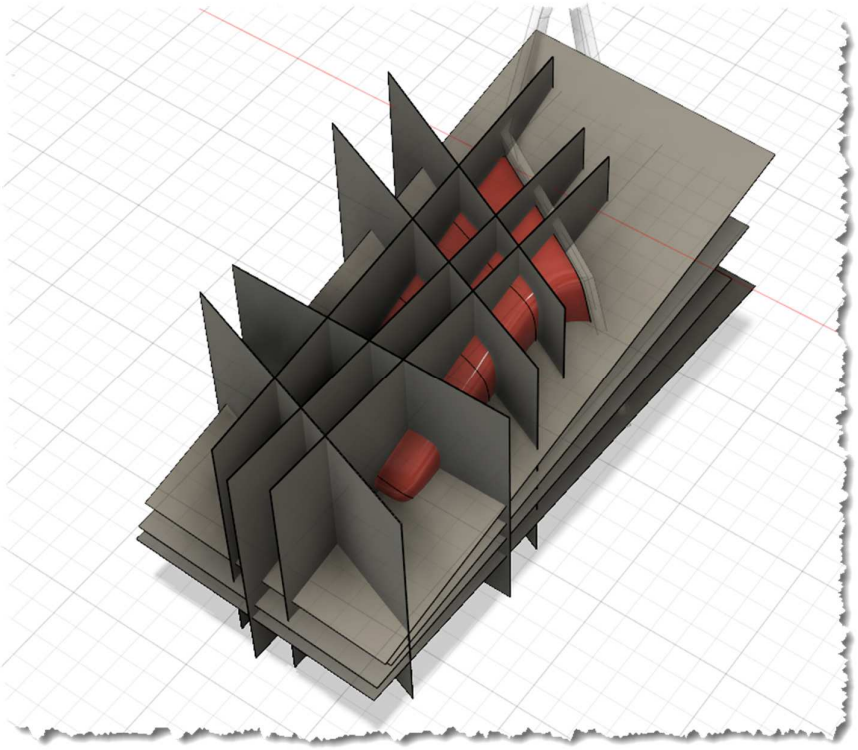


12. From the project tree, select the original base geometry and deleted it since it is not going to be used anymore (right click to visualize the contextual menu);

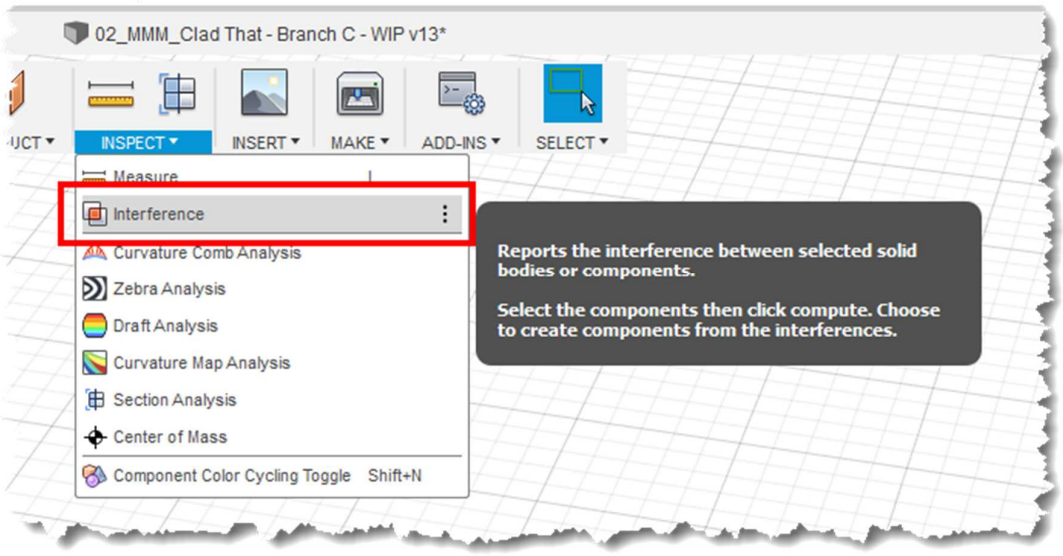


13. Click on “Finish Form” and repeat the procedure followed on the previous exercise to thicken the shell and divide it in panels;

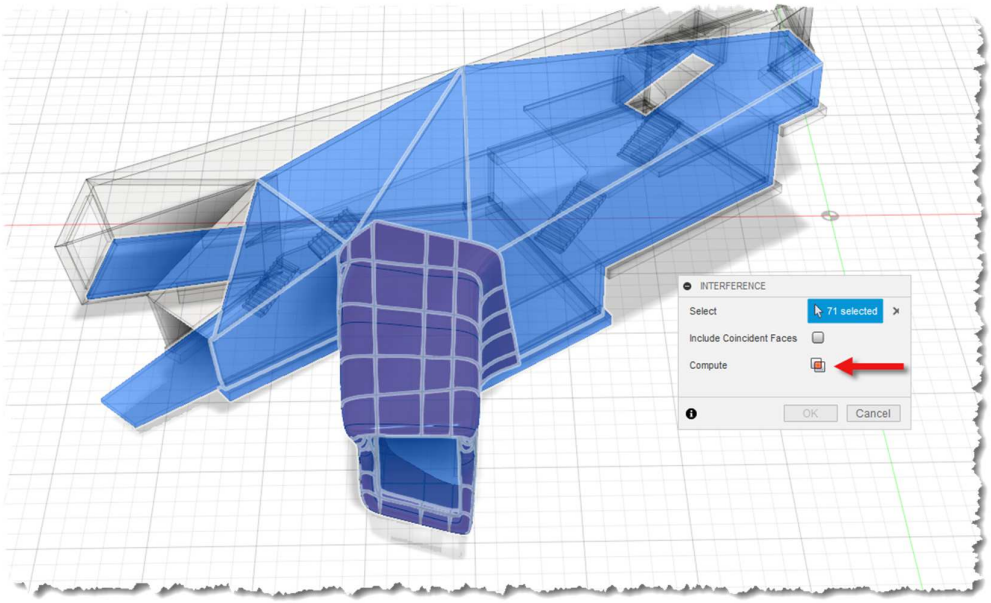




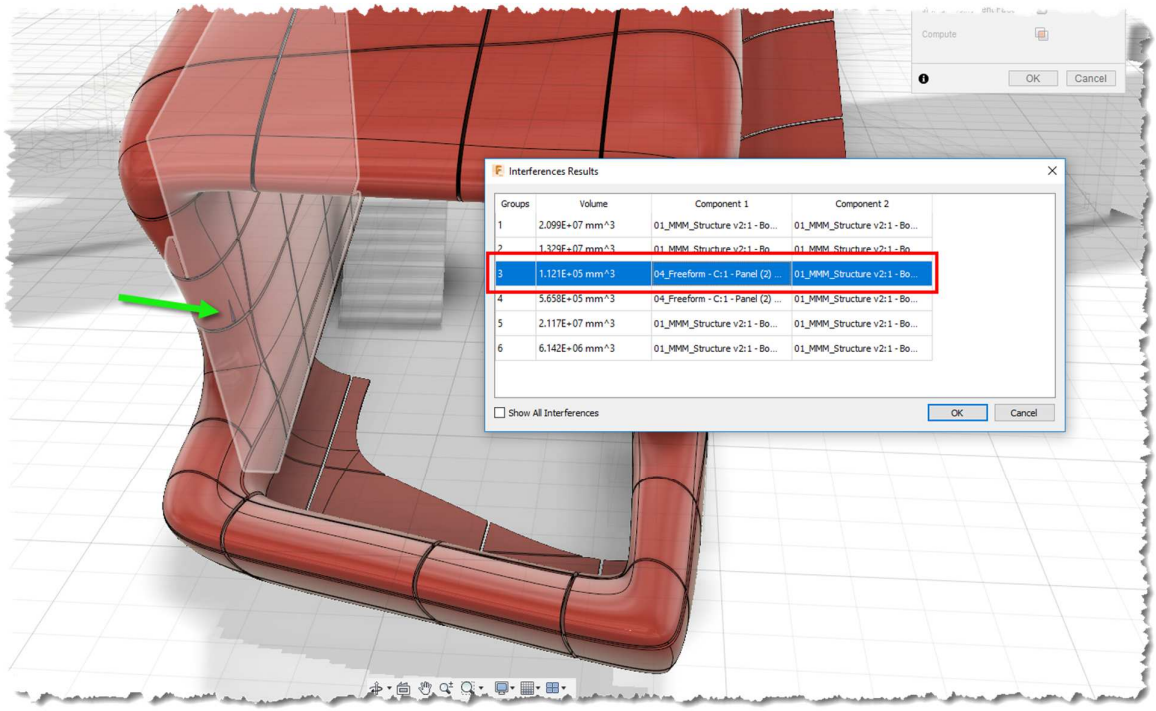
14. When solid models have been created, it is possible to the if there are interferences between geometries; use the “Interference” command under the “Inspect” menu;



15. Select the elements to test;

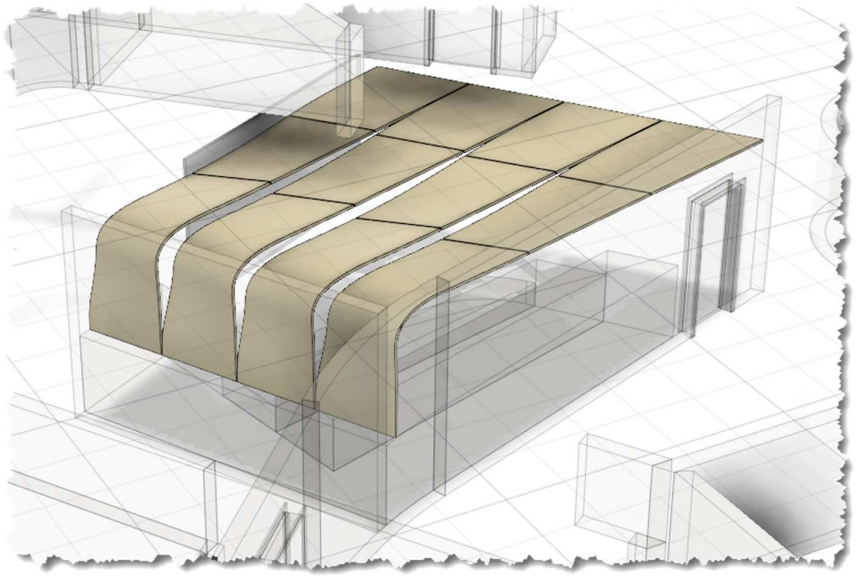


16. Check the results and correct the found issues;



11.5 Theater acoustic ceiling

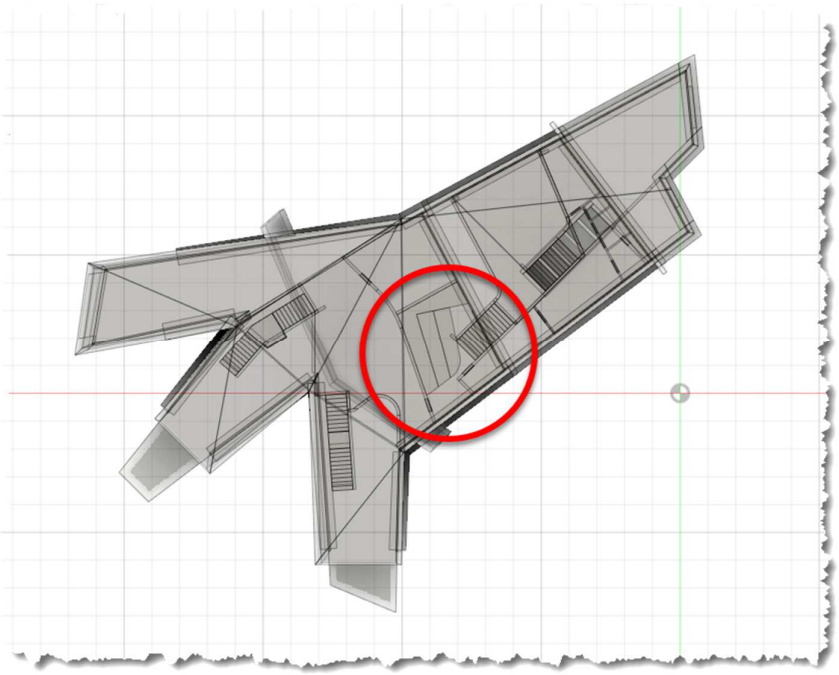
The scope of this exercise is showing how an interior cladding can be created for a bounded space. This is a small cinema on the lower floor of the building.



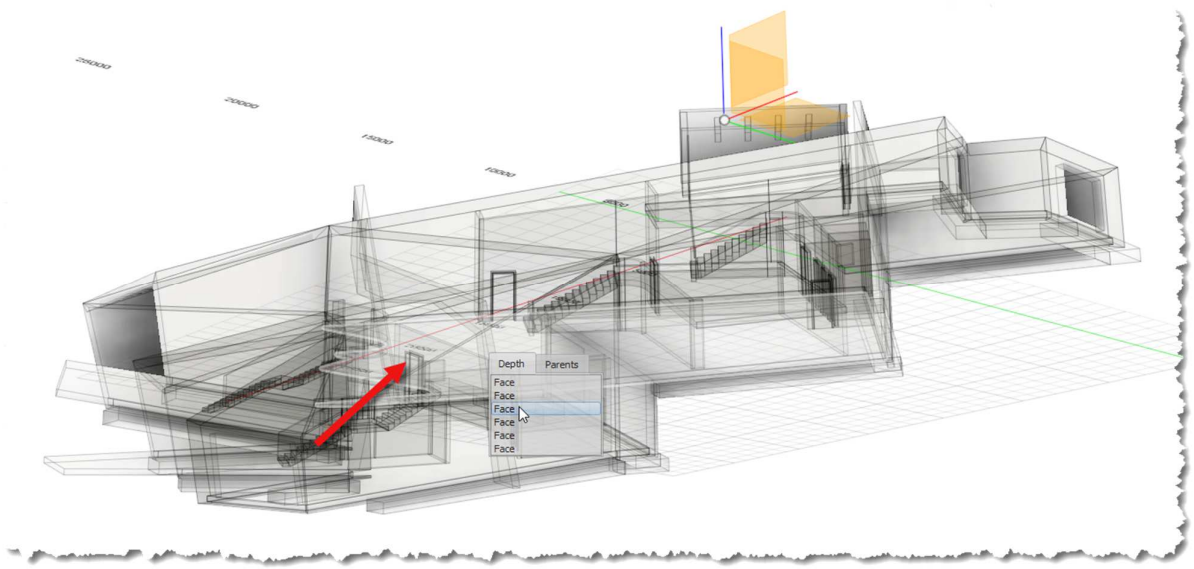
Use the file called “03_MMM_Clad That - Cinema T.F3D” to start a new design or open it from the A360 project.

11.5.1 Ceiling: Main shape creation

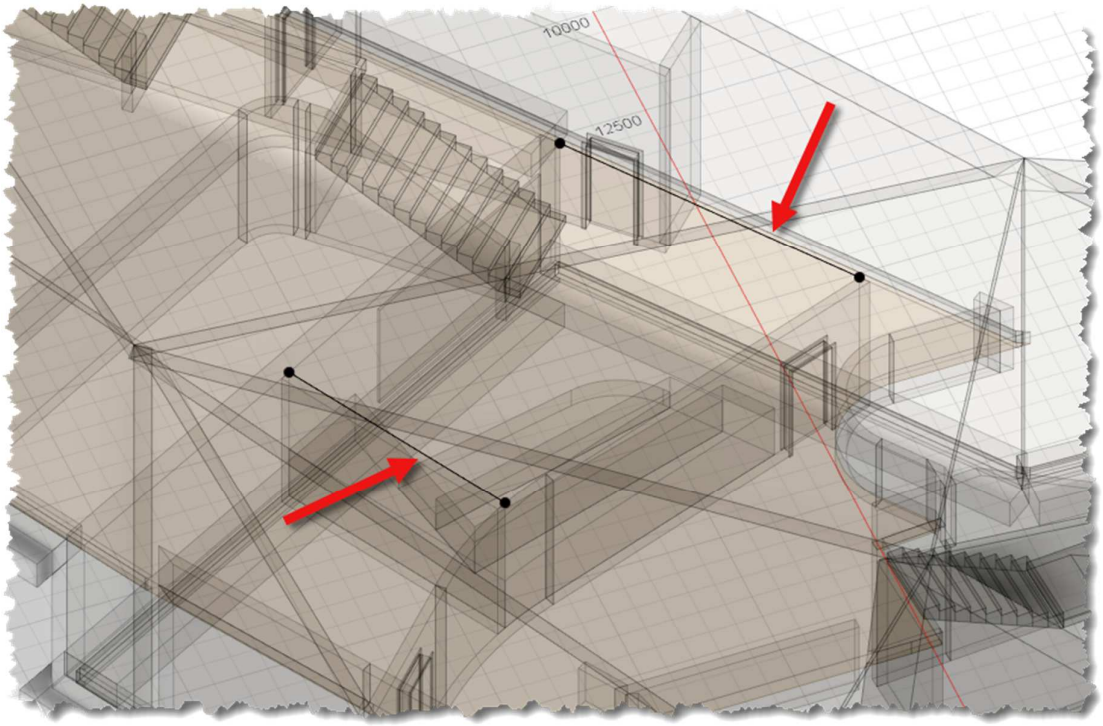
1. Create a new component called “04_Cinema”;
2. Following image shows the building area in which the acoustic ceiling must be modeled;



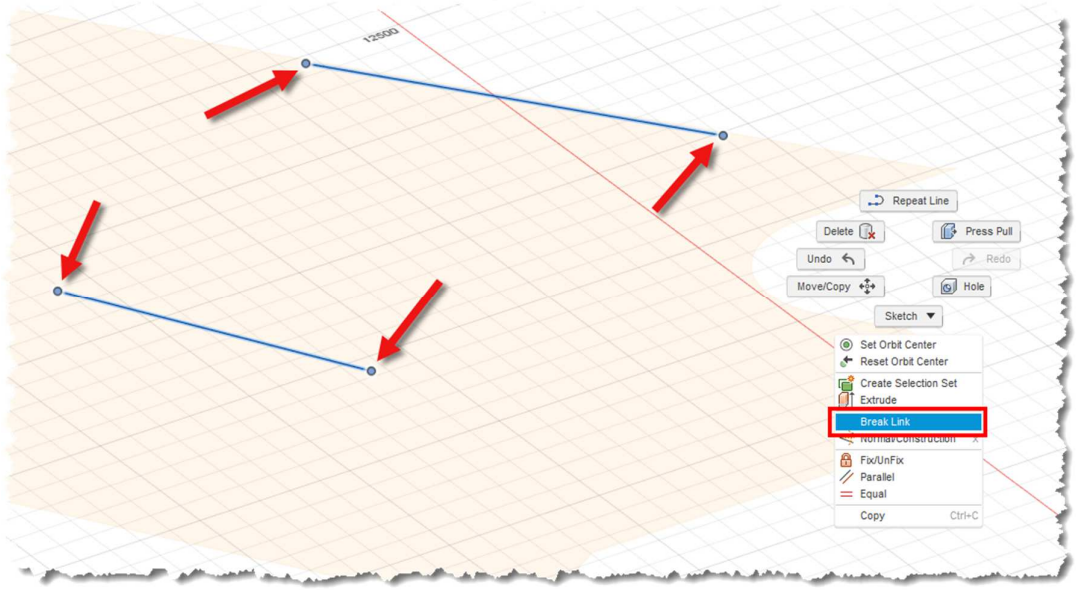
3. Orient the view to clearly see the bottom face of the second structural slab;
4. Start a new sketch and indicate that face as the host; keep the left mouse button pressed to have Fusion 360 showing a dropdown menu allowing to choose the right host;



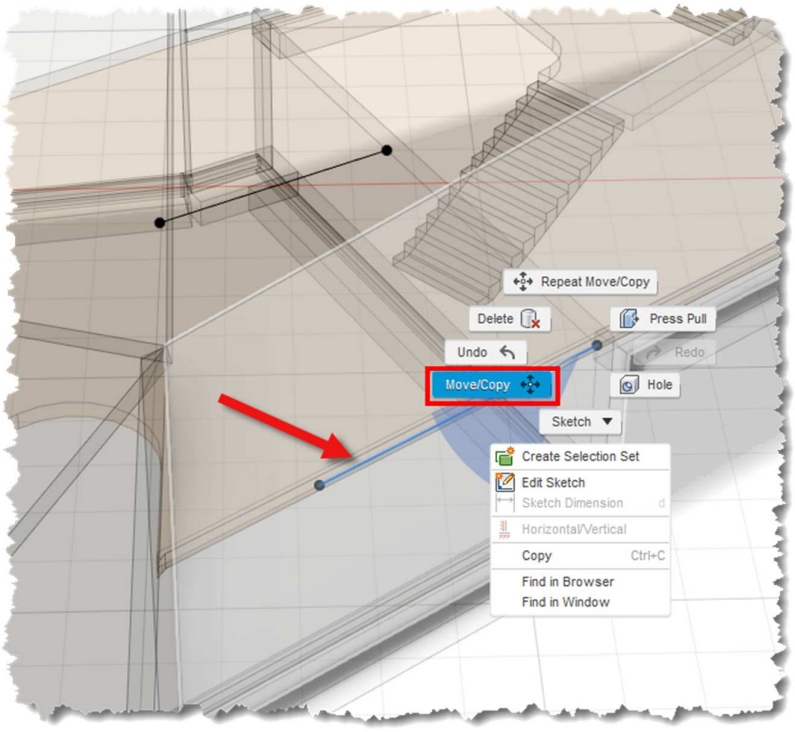
5. Create a sketch using as a reference the walls edges; if needed, use the “Project” command; two lines only are needed in this case;



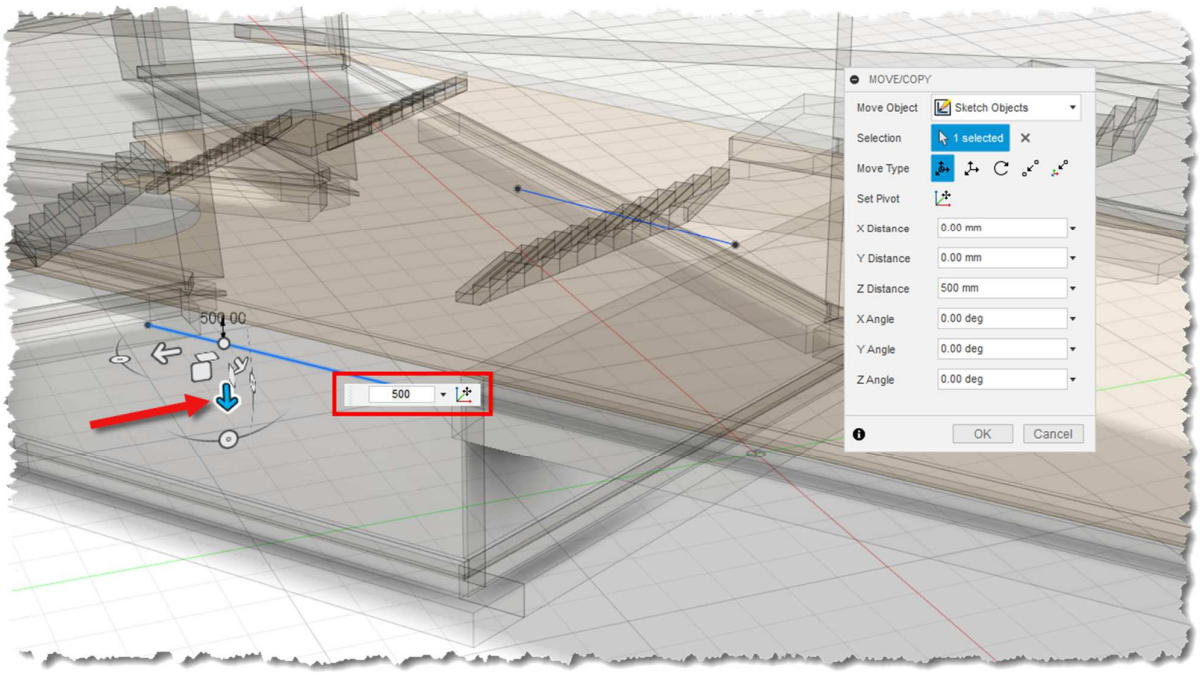
6. Select all the sketch elements, right click and choose “Break Link” from the contextual menu to remove the connection with the original projected geometry. This allows to freely manipulate the sketch elements;



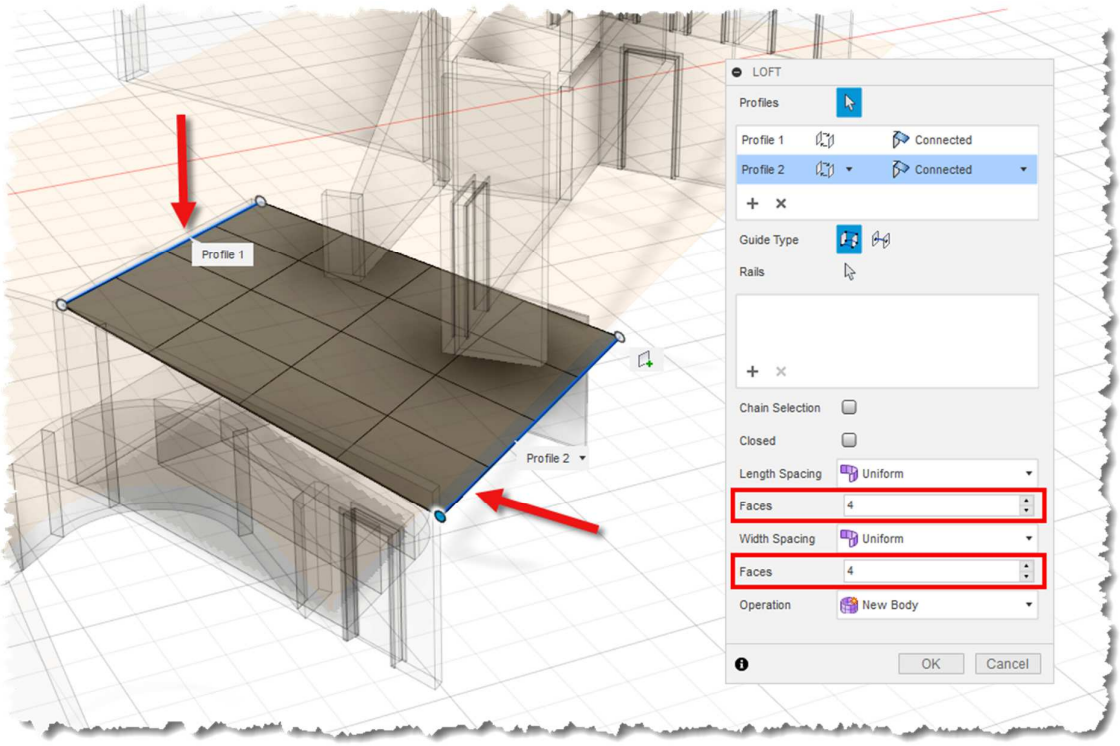
7. Finish the sketch;
8. Select the outer line and make right click; choose the “Move/Copy” command;



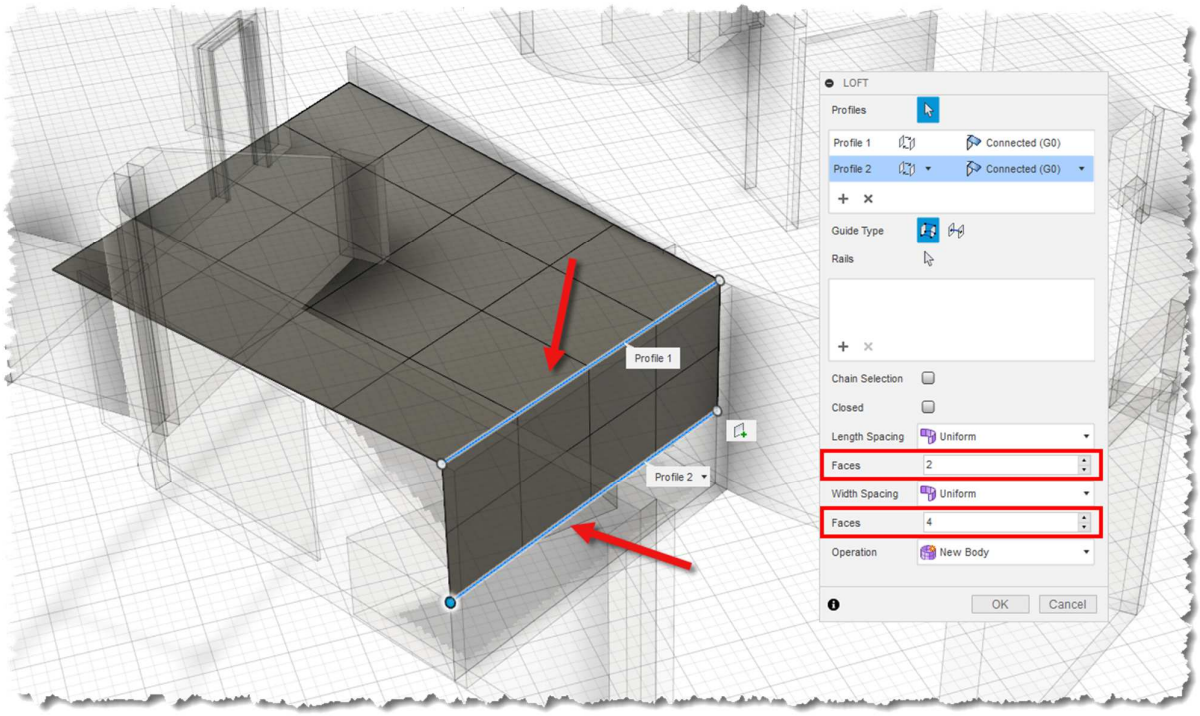
9. Move the line downwards by 500 mm and click “Ok”;



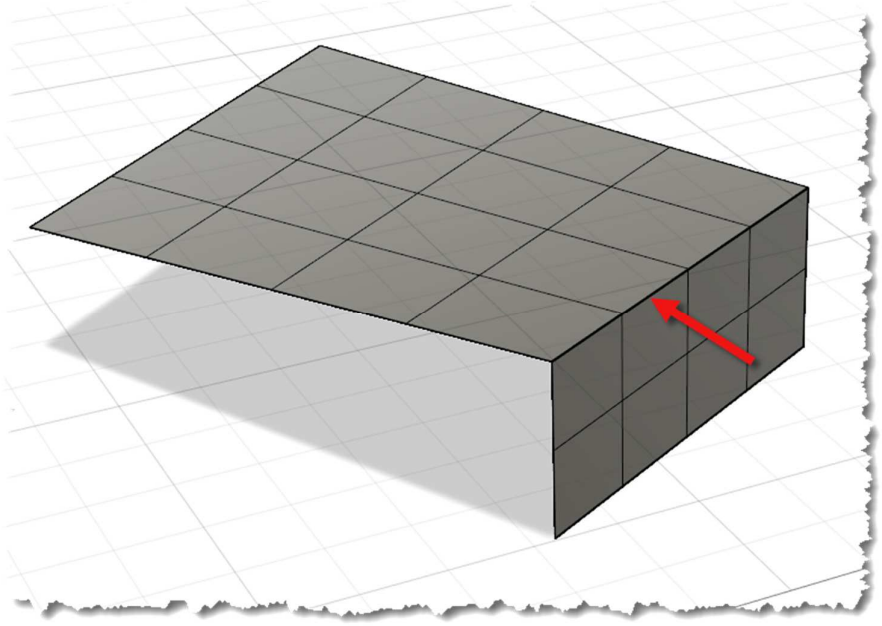
10. Click on “Create Form” to enter the freeform modeling workspace;
11. Create a loft using the two lines as profiles and specify 4 faces both for Length and for Width directions;



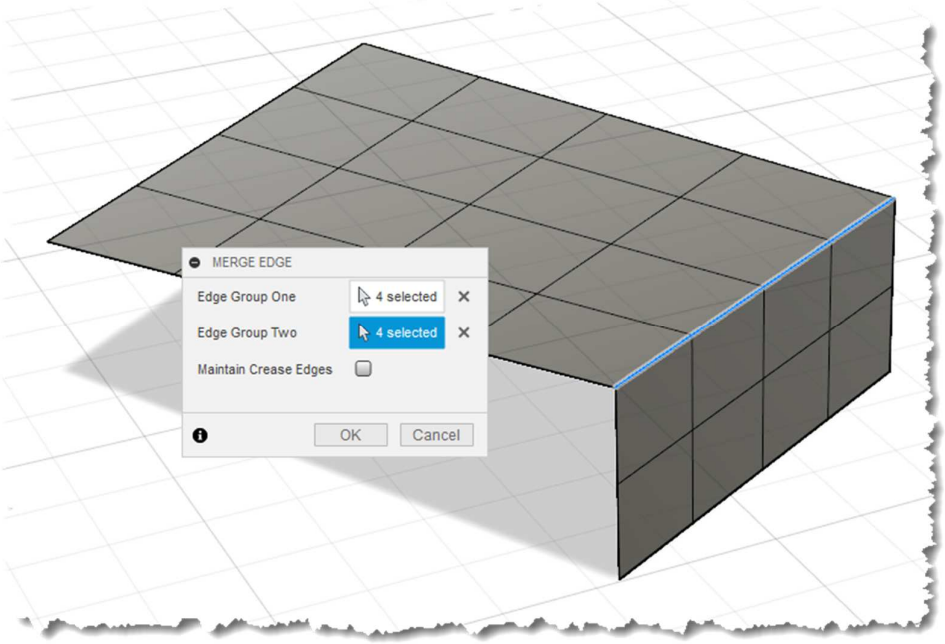
12. Create another Loft, between the ceiling edge and the seat element edge as in the following picture; divide the surface with 4 and 2 faces as shown in the Loft dialog below;



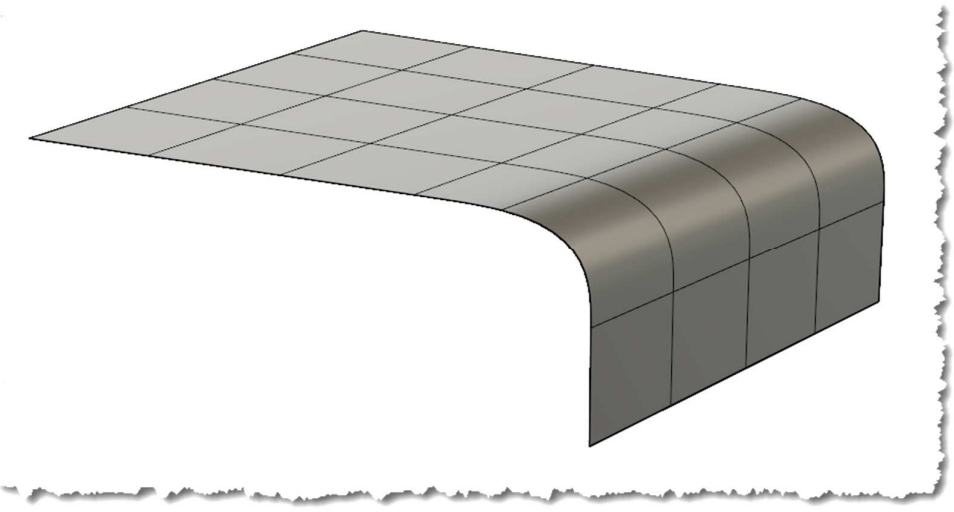
13. Click “Ok” when done;
14. The edges must be merged to obtain a smooth transition and a single surface;



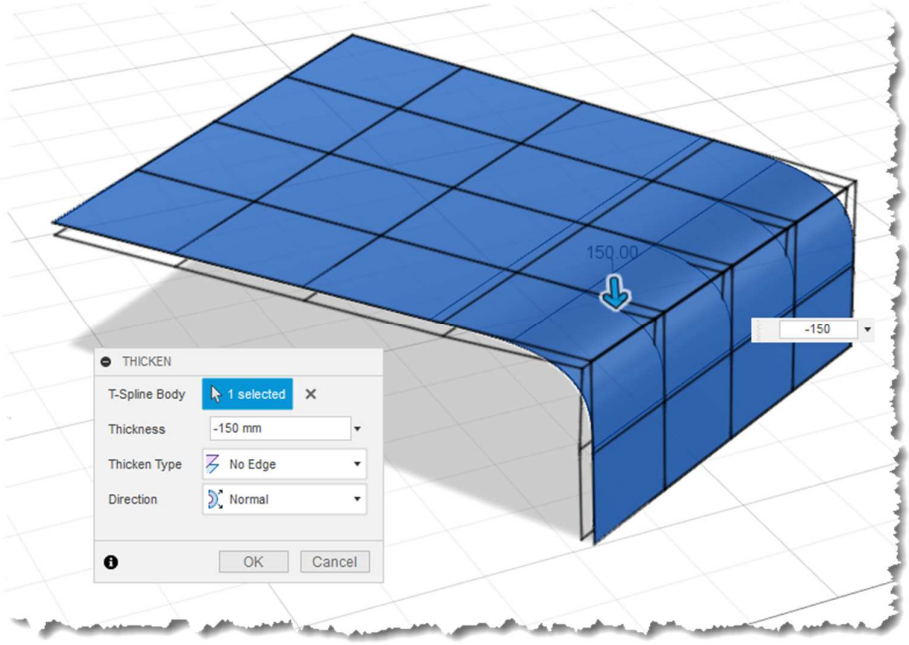
15. Use the “Merge Edges” command;



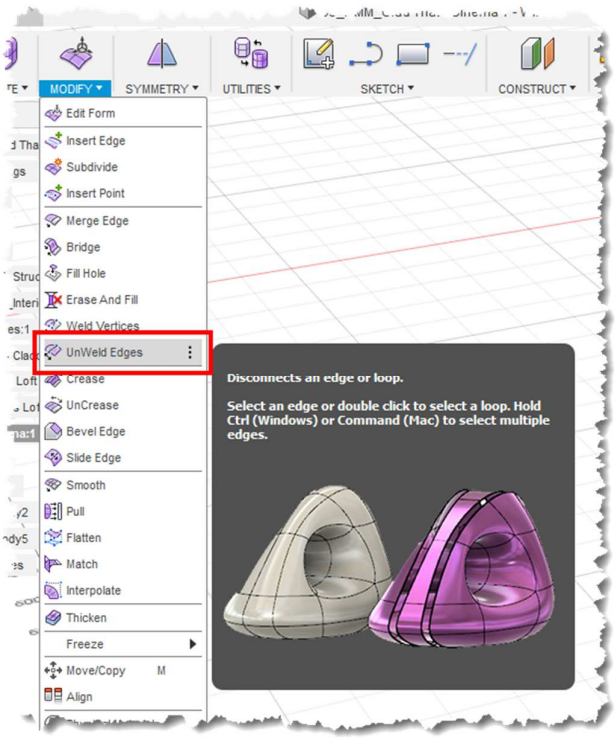
16. The results should be like that one in the next image;



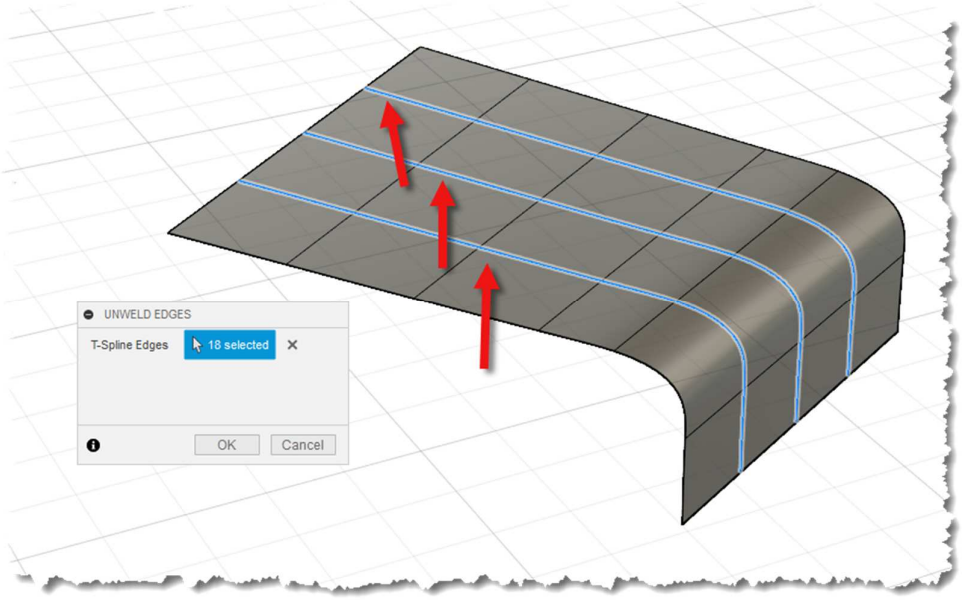
17. Use the “Thicken” command to create and offset of the surface downwards; specify 150 mm of thickness but check the direction before finalizing the command. Specify “No Edge” as “Thicken Type”;



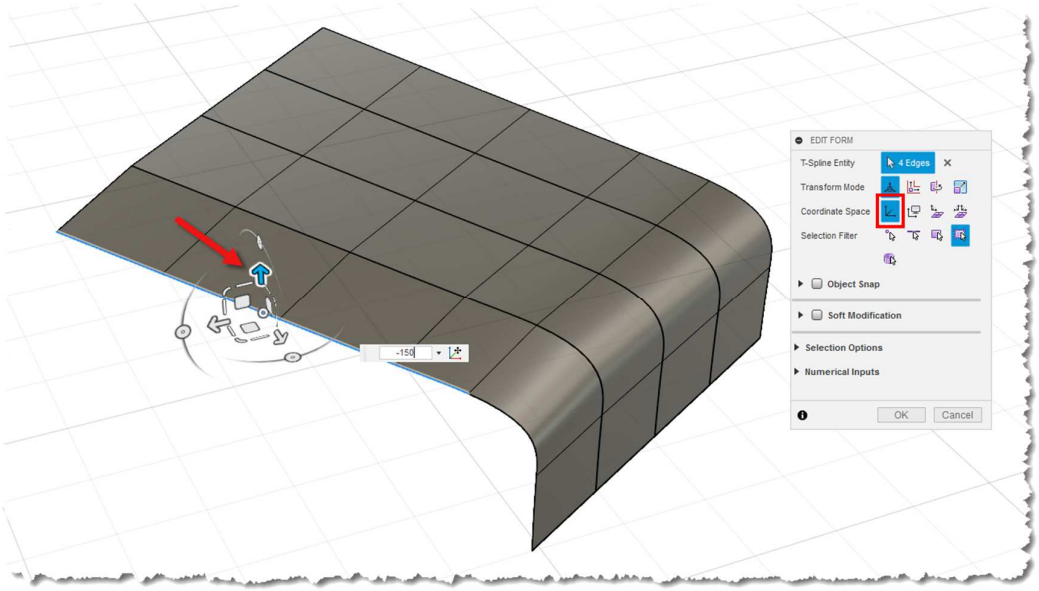
18. It is now necessary to divide the surface in 4 different “stripes” so that it will be possible to manipulate the shape of each one easily;
19. Click on the “UnWeld Edges” command;



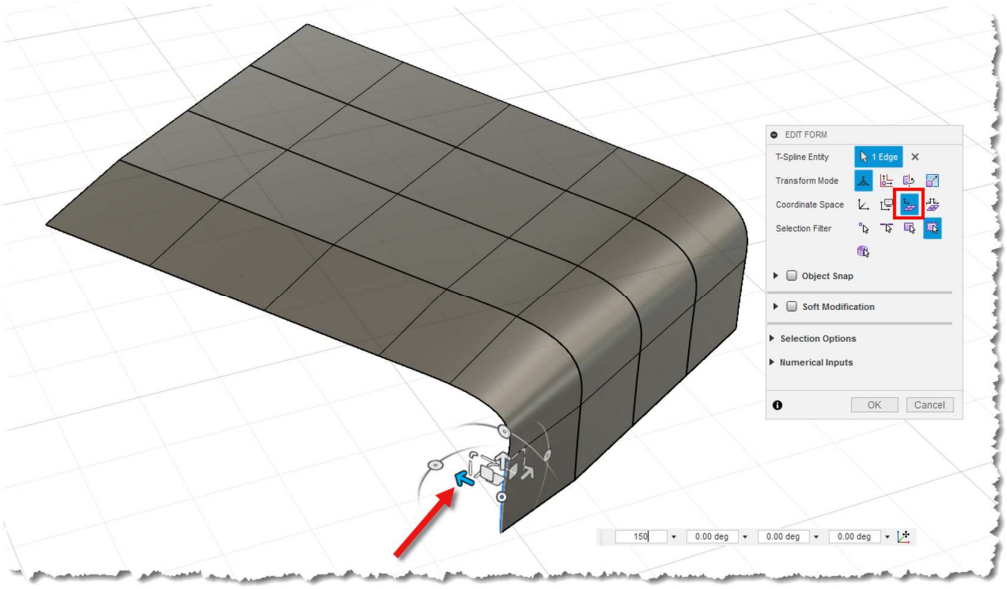
17. Select the highlighted edges keeping pressed the Shift key to add elements to the current selection. It is also possible to double click on a line to select the whole chain. Click “Ok” when done;



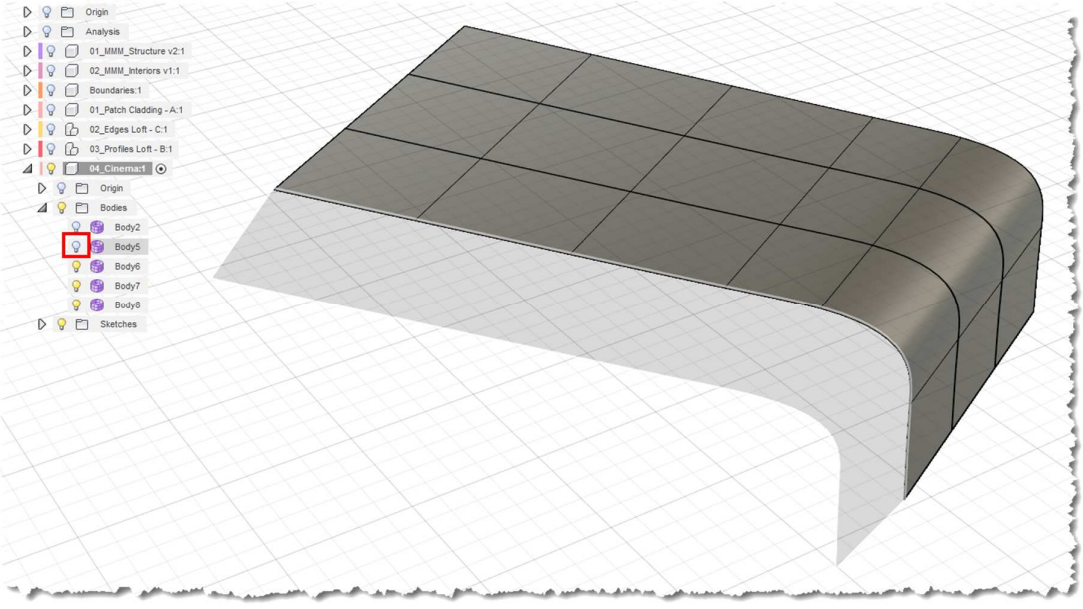
18. The surface has been split in four different bodies;
19. Select the highlighted edges (four elements in this case) and use the “Edit Form” command to move them downward by 150 mm;



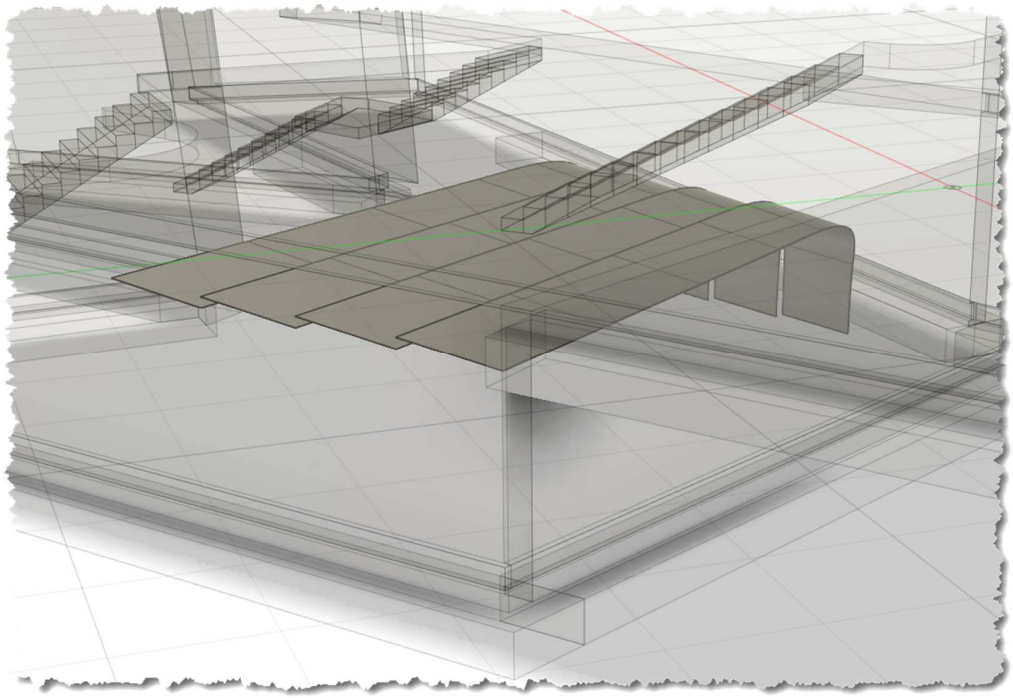
17. Select the vertical edge and move it to left for 150 mm



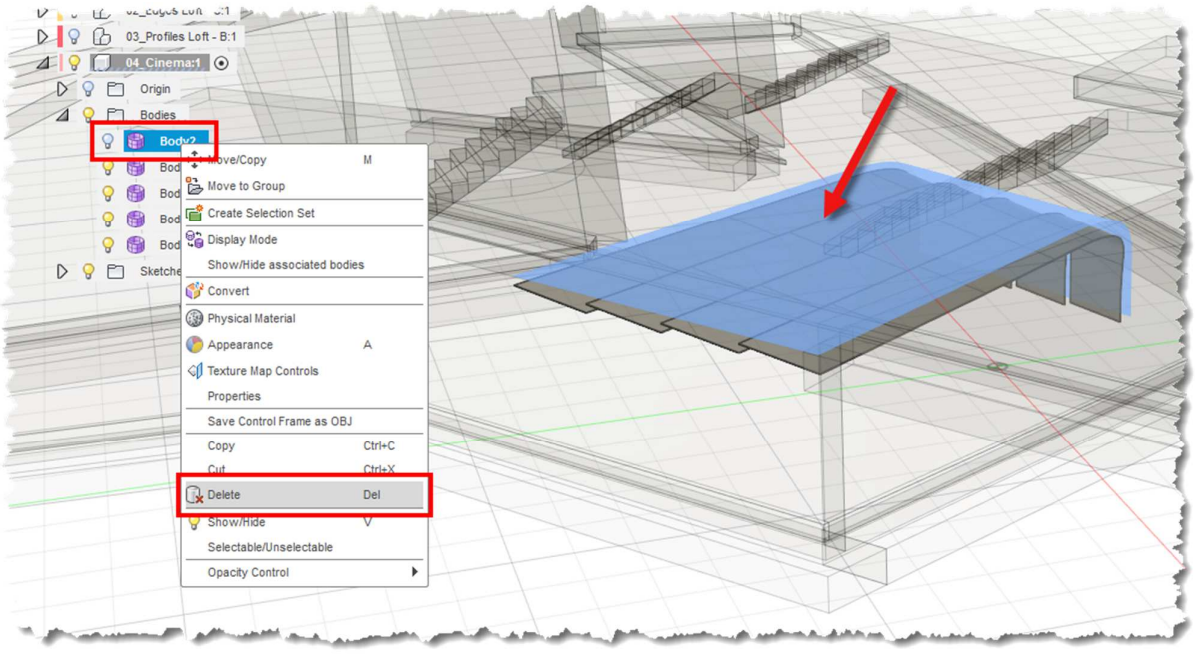
18. Switch off the visibility of the manipulated body and repeat the same set of operations on the following set of edges;



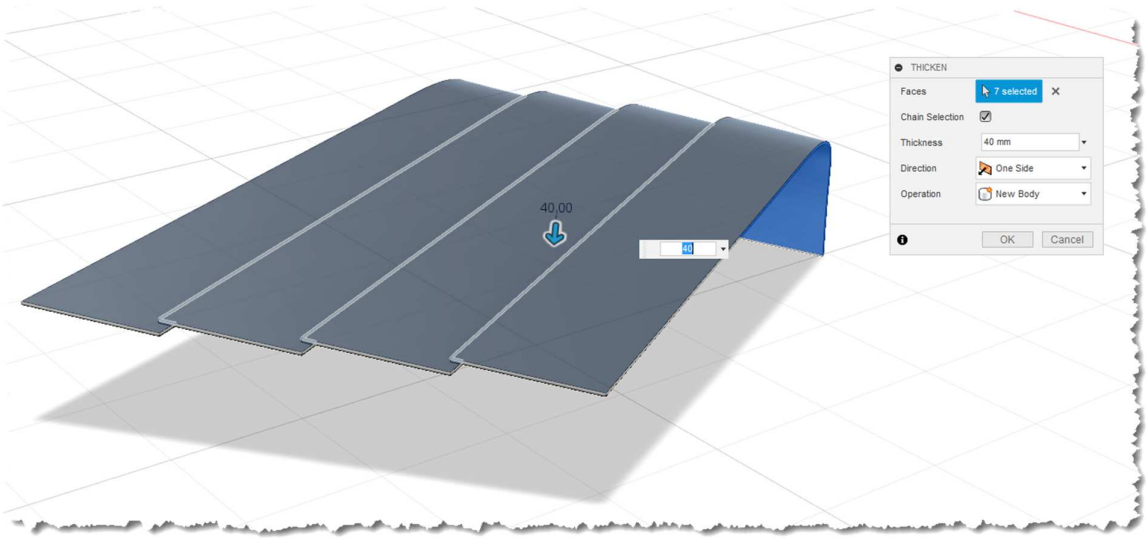
19. Repeat the same procedure on the remaining two bodies; the result should look like to the following picture;



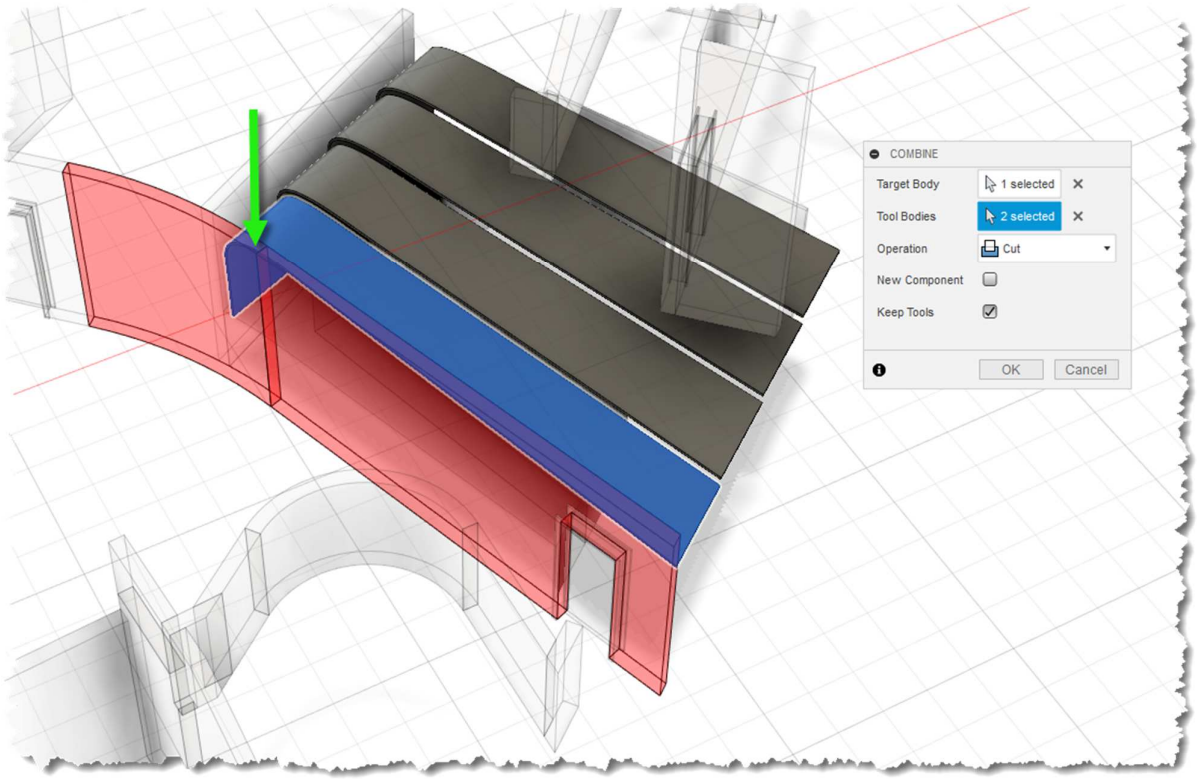
20. Remove the basic shape since it is not needed anymore;



21. Use the “Thicken” command (40 mm downward) to create solid bodies out of the four surfaces;



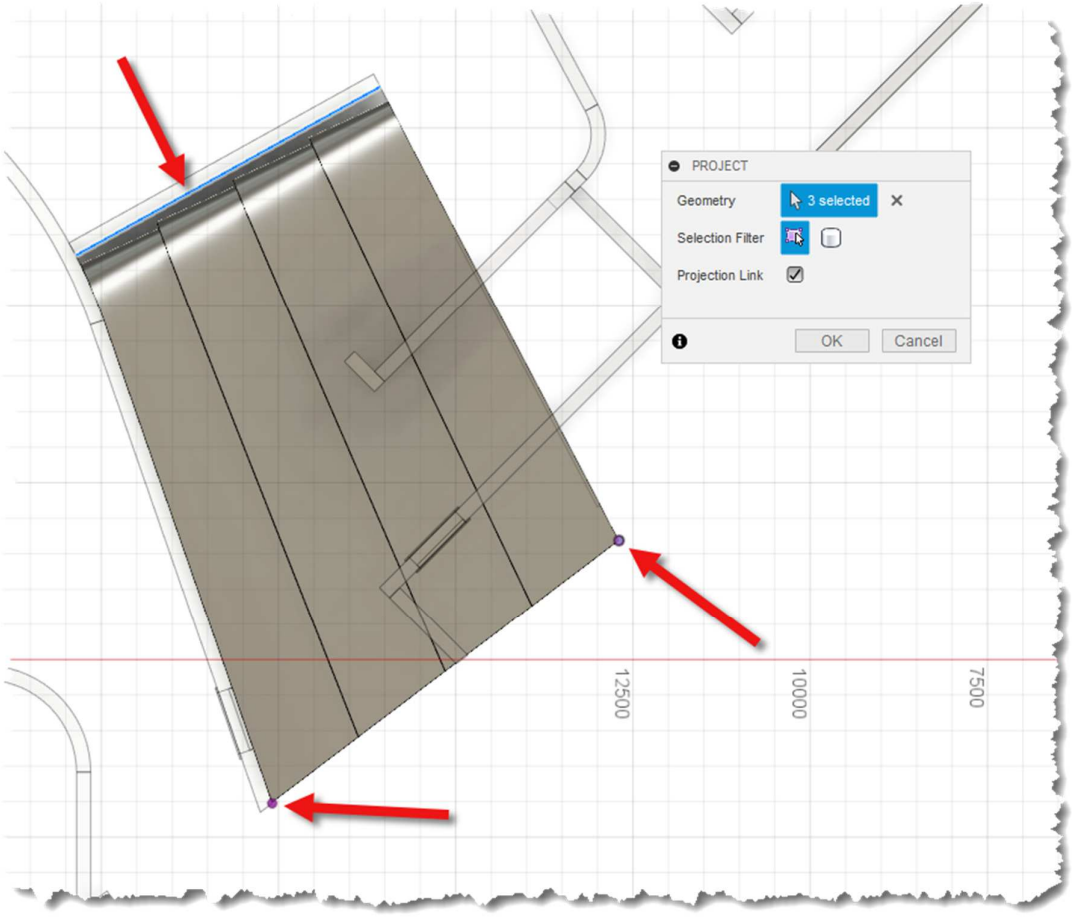
22. Cut the element closer to the entrance using the wall as the tool. This cut is needed since that wall has a curved part which interferes with the ceiling component;
23. Remember to specify “Keep Tools” in the “Combine” options;



11.5.2 Ceiling: panels division

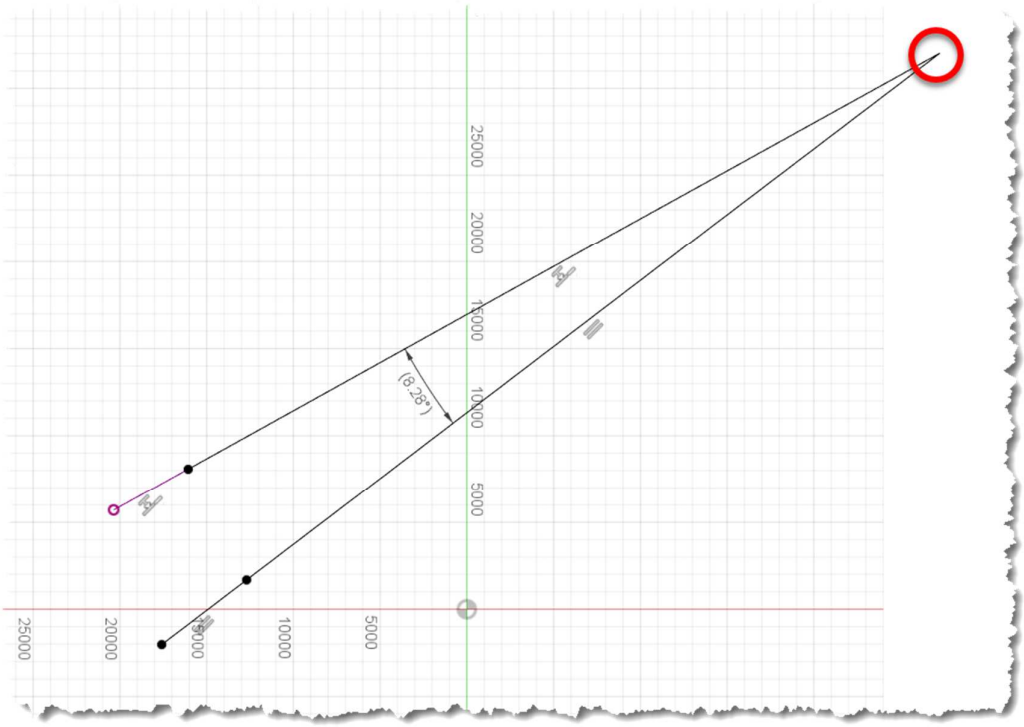
The scope of this exercise is understanding the method for dividing the ceiling in three parts with the same angle.

1. Start a new sketch and use the XY plane as the host;
2. Use the “Project” command to capture the wall projection and the two points identified by the arrows in the image below;

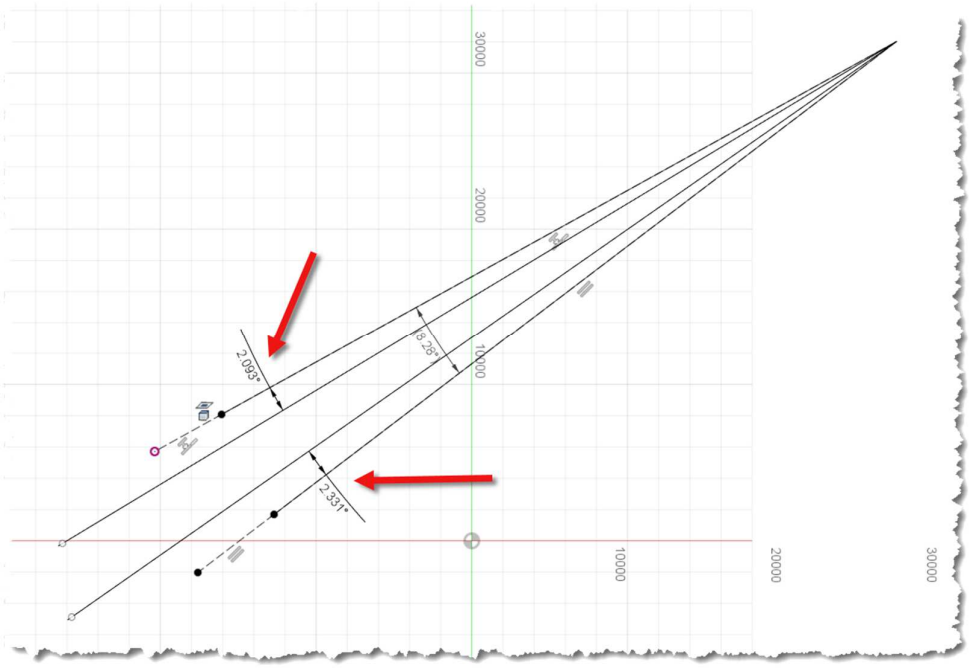


3. Create a line between the two points and extend both the lines to reach their intersection point;

4. Place an angular dimension to measure the angle between the two lines;

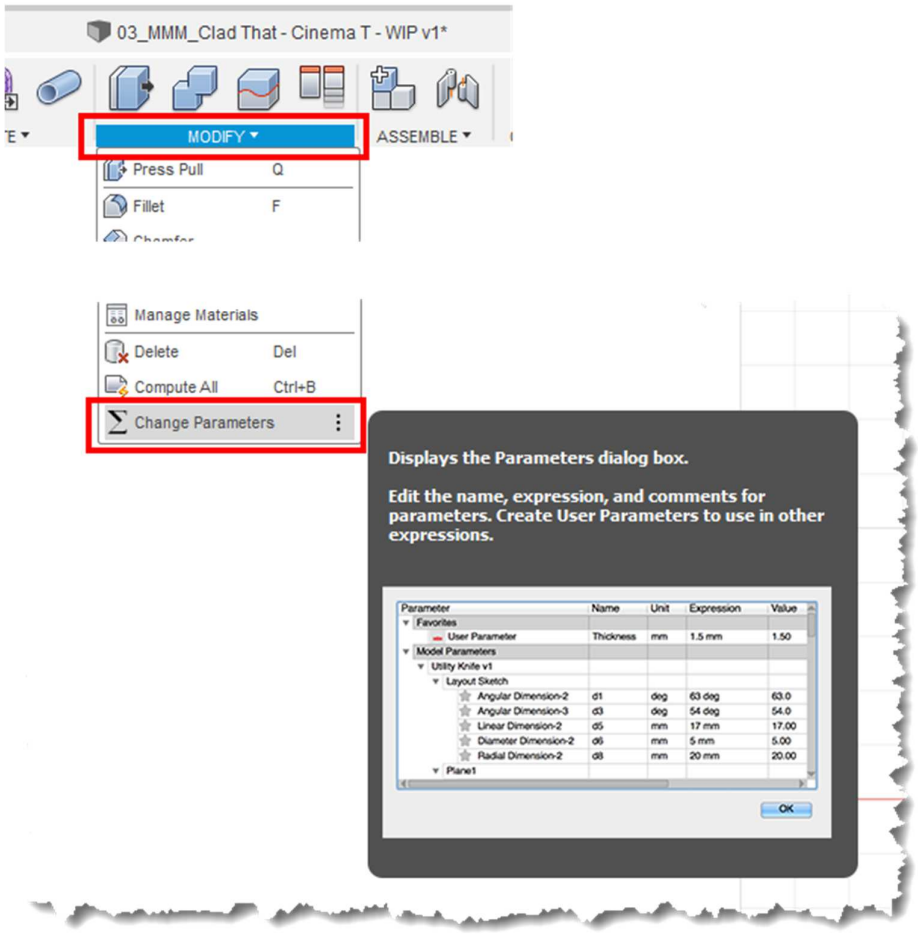


5. Create two additional lines inside the previous two ones and place and angular dimension for each of them;

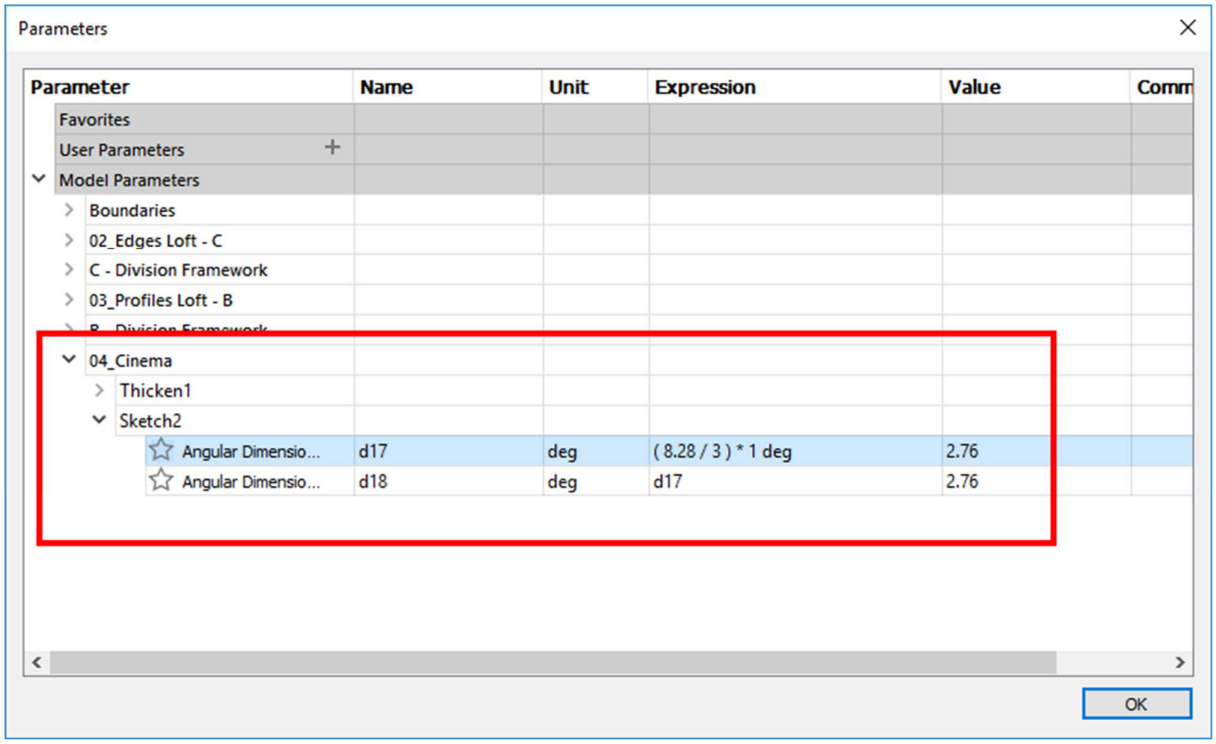


6.

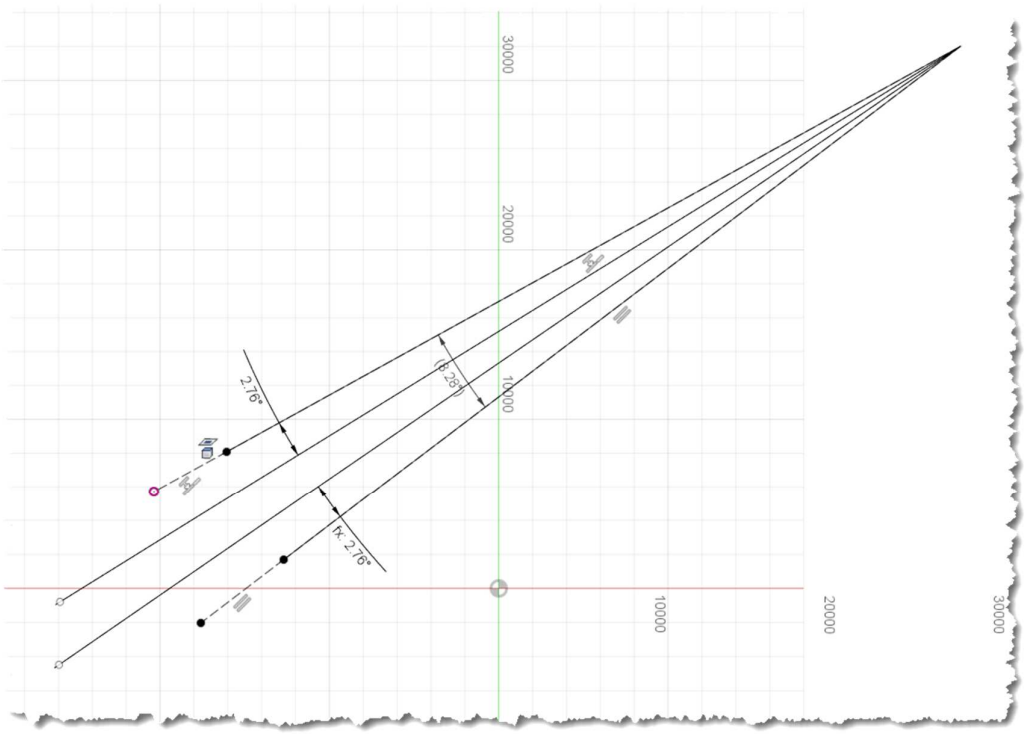
6. Open the “Change Parameters” dialog; the command is under the “Modify” menu of the “Model” workspace;



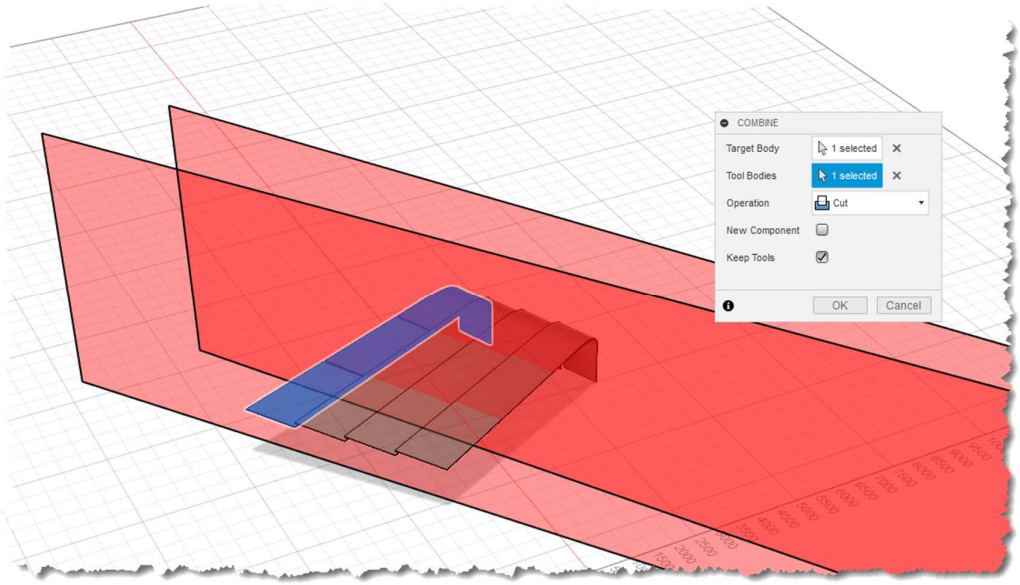
7. This dialog allows to list the properties of each element; elements are sorted and organized by component. Expand the “04_Cinema” component and look for the sketch. Once it has been found, set the first angular dimension as the measured angle divided by three (3) under the “Expression” column. Set as the expression of the second angular dimension the name of the first one (in this case: **d17**). This forces the first dimension to the one third of the main angle and the second dimension to acquire the same value of the first one.



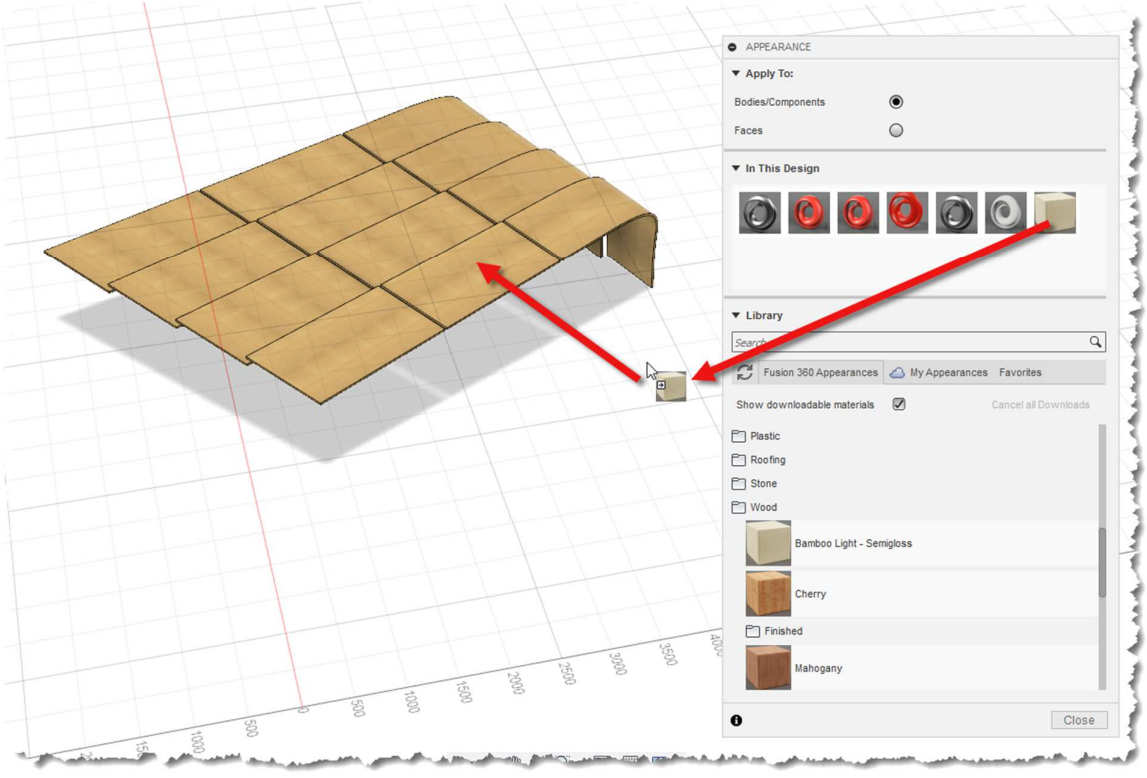
8. The result is that the three lines are equally spaced in terms of angle width;



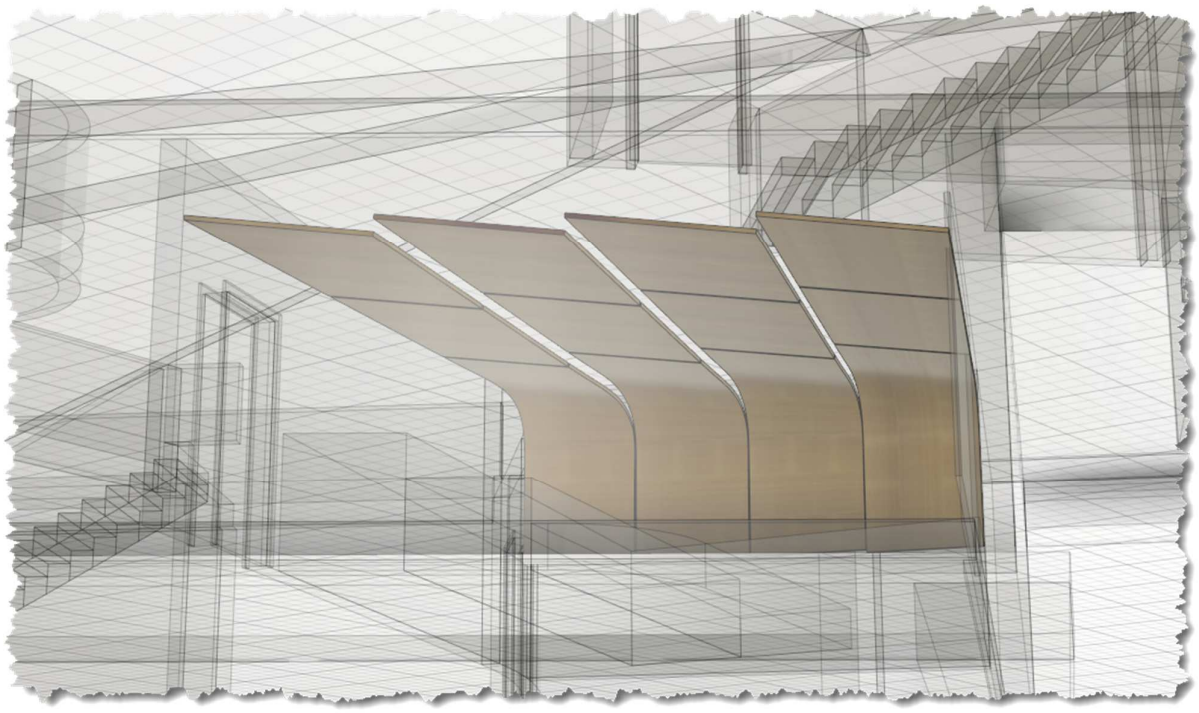
9. This sketch must be used to extrude three vertical surfaces (using the Patch environment, as seen previously in this document). These surfaces are then thickened and used to cut the ceiling bodies;



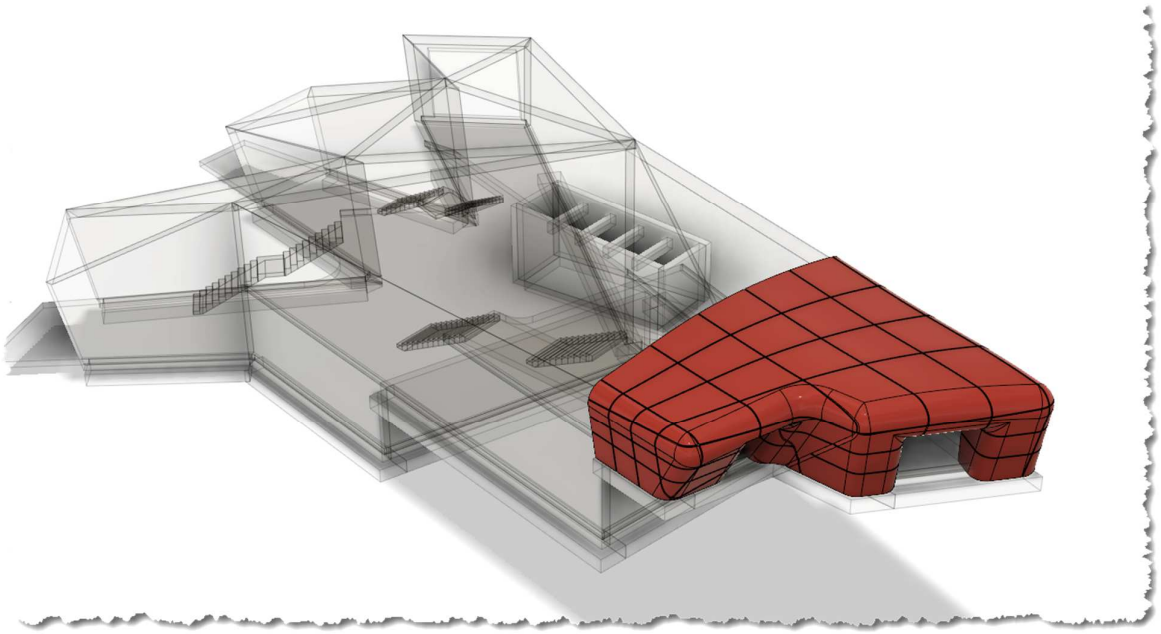
10. If desired, load or create a material in the "Appearance" dialog and apply it to the ceiling elements by using the drag & drop technique. In this case a wood material has been used;



11. Next picture shows the result of this exercise inside the building;



12 Exercise #4: Solid Modeling – the Entrance

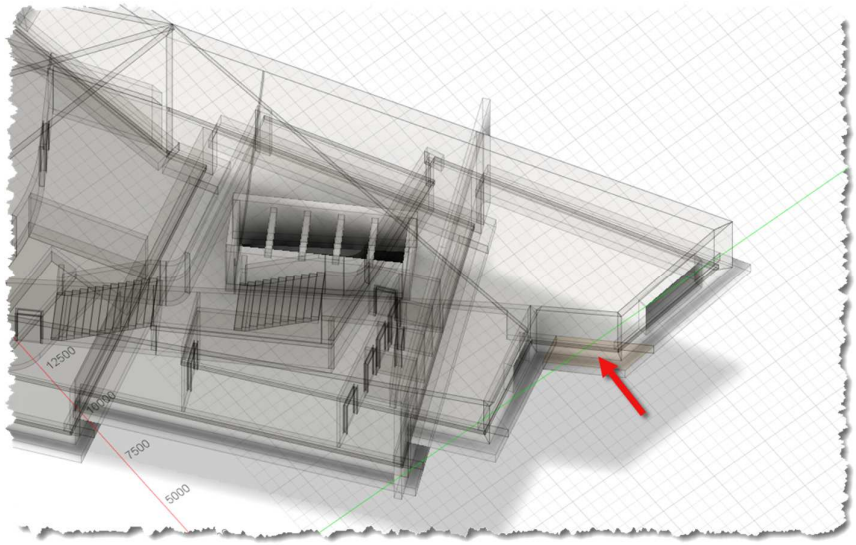


The scope of this exercise is using the solid modeling environment to define the exterior shell of building entrance area. This volume will be then “carved” to create the cladding panels.

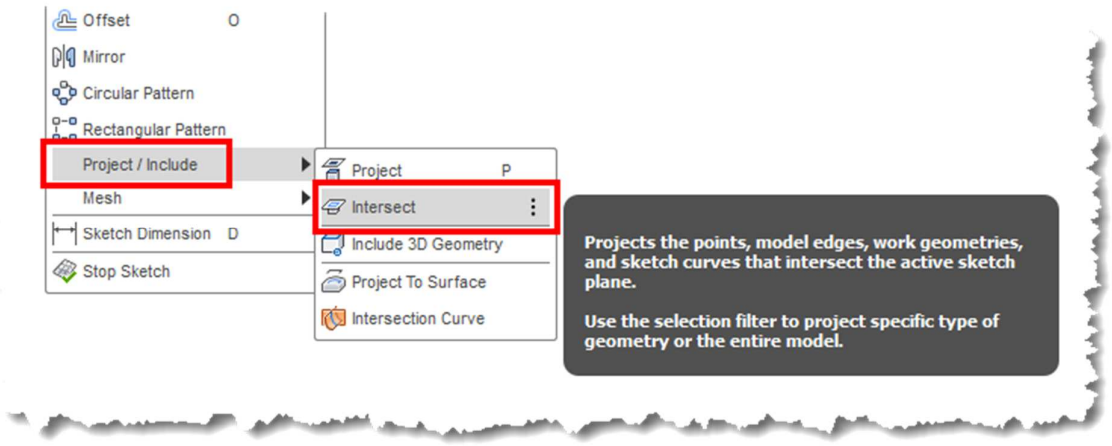
This is another method which can be used to create the cladding for a building, so the user can choose whether applying the techniques related to the Patch environment (the standard surface modeling), the Freeform area (the Sculpt environment) and the Solid Modeling. The three approaches can be combined is needed to expand the designer’s capabilities.

12.1 Envelope volume definition

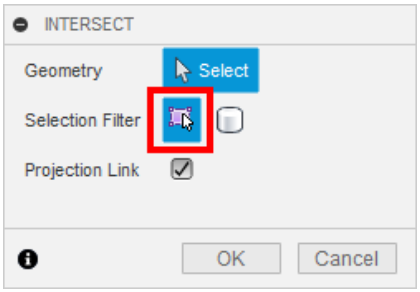
1. Create a new component, called “05_Entrance” and activate it;
2. Start a new sketch using the top face of the continuous foundation as the host;



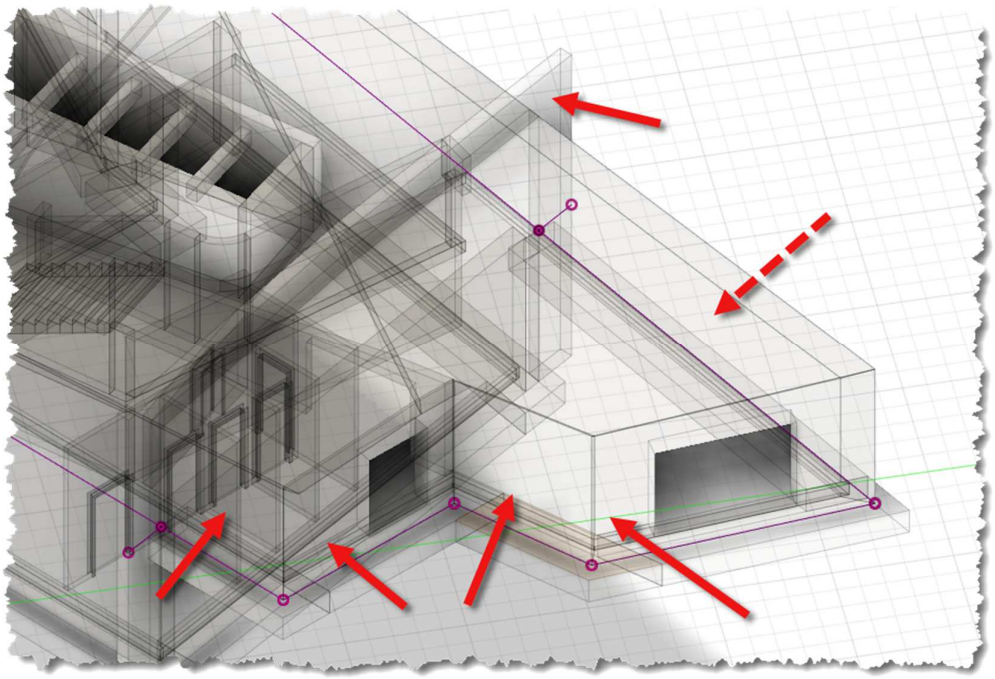
3. Use the “Intersect” command, under the “Sketch” menu, to include in the sketch the intersection of selected geometries with the host plane; it is possible to select either faces or bodies;



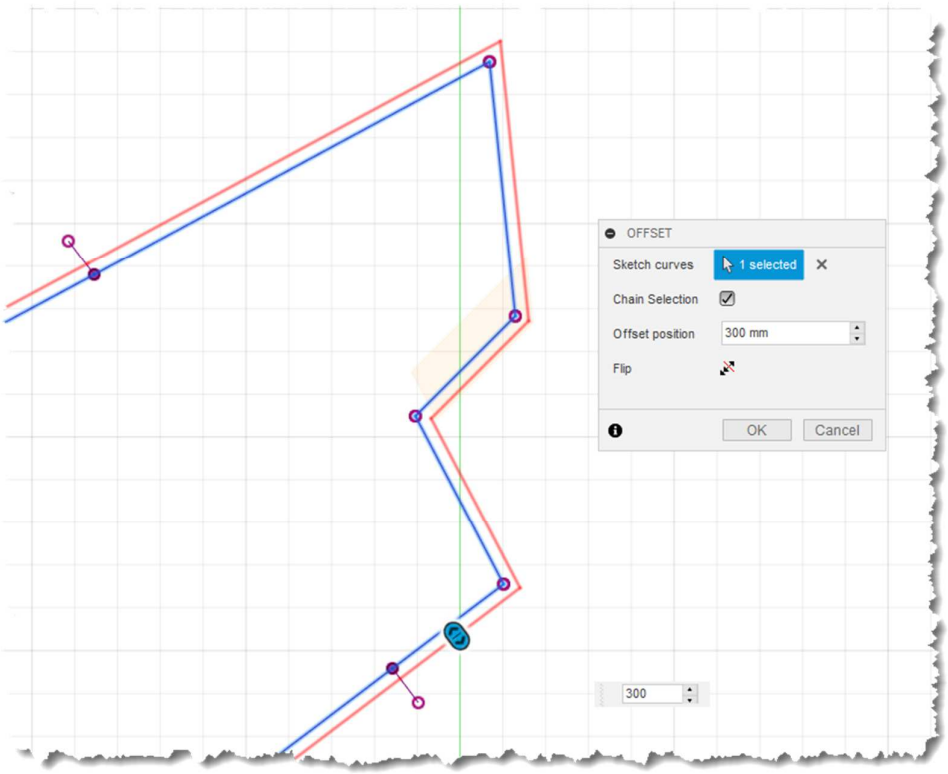
4. Specify “Faces” as “Selection Filter”;



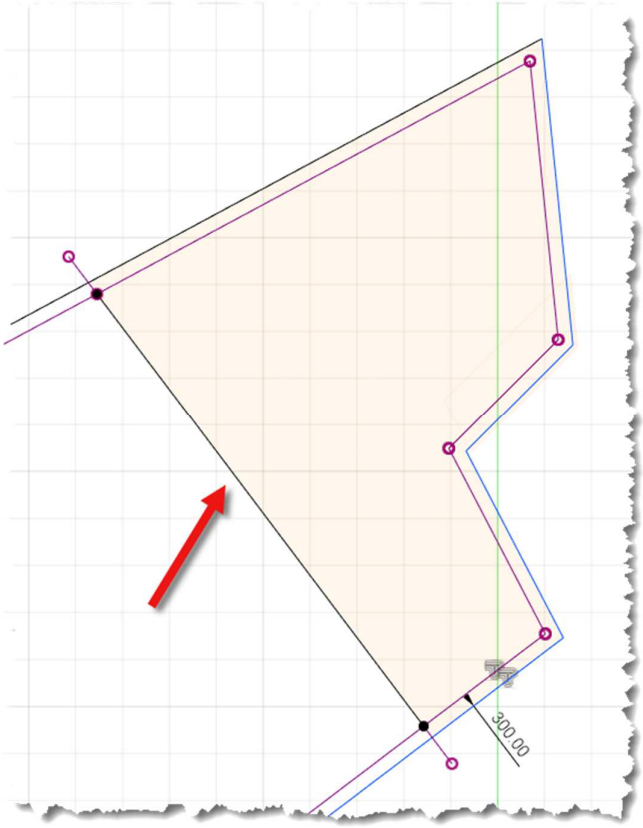
5. Select the faces highlighted in the picture below; a purple line will appear when an intersecting face is selected; Click “Ok” when done;



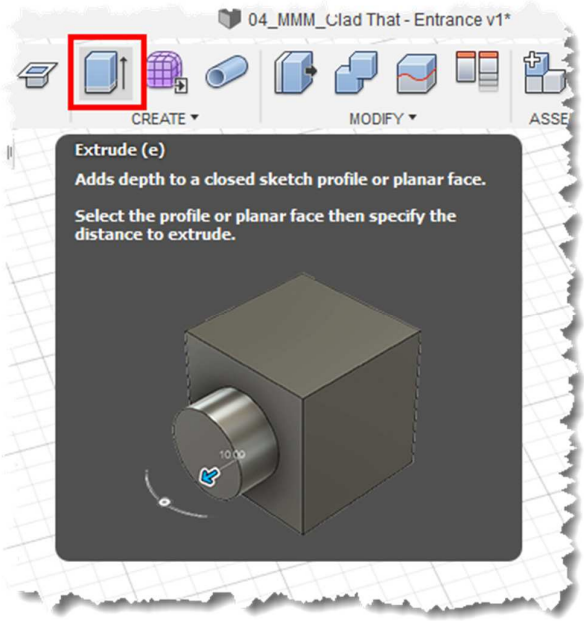
6. Make an offset outward of 300 mm;



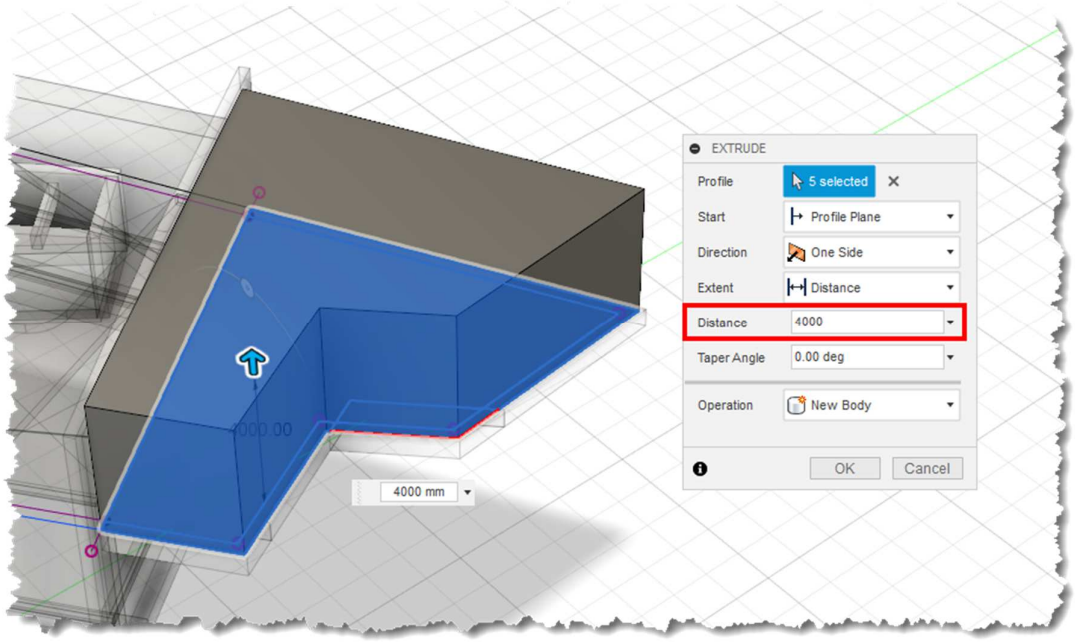
7. Add the internal line as show below to create a closed profile;



8. Use the “Extrude” command under the “Create” menu of the “Model” area;



9. Click on the “Extrude” command and select all the profiles in the sketch created previously. Set 4000 mm as the extrusion “Distance”;

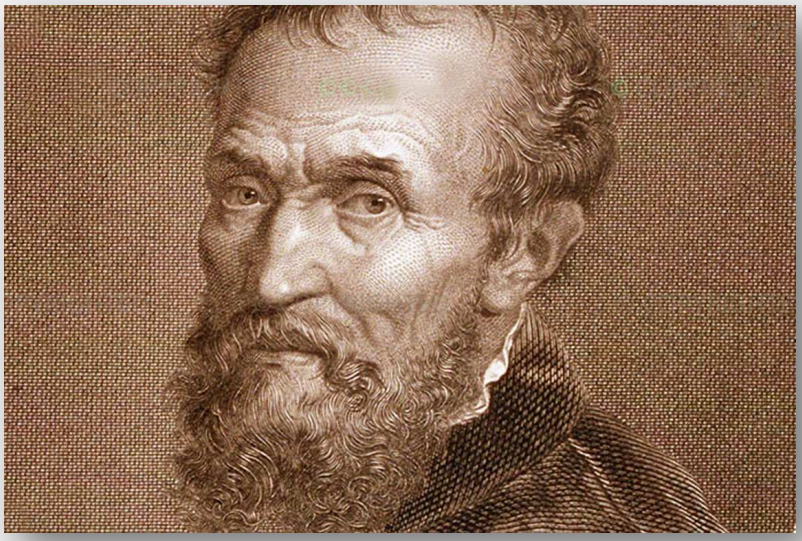


10. The created solid is the “container” of all the geometries defining the cladding; this has to be “carved” to remove the excess of material and leave the desired shape only.

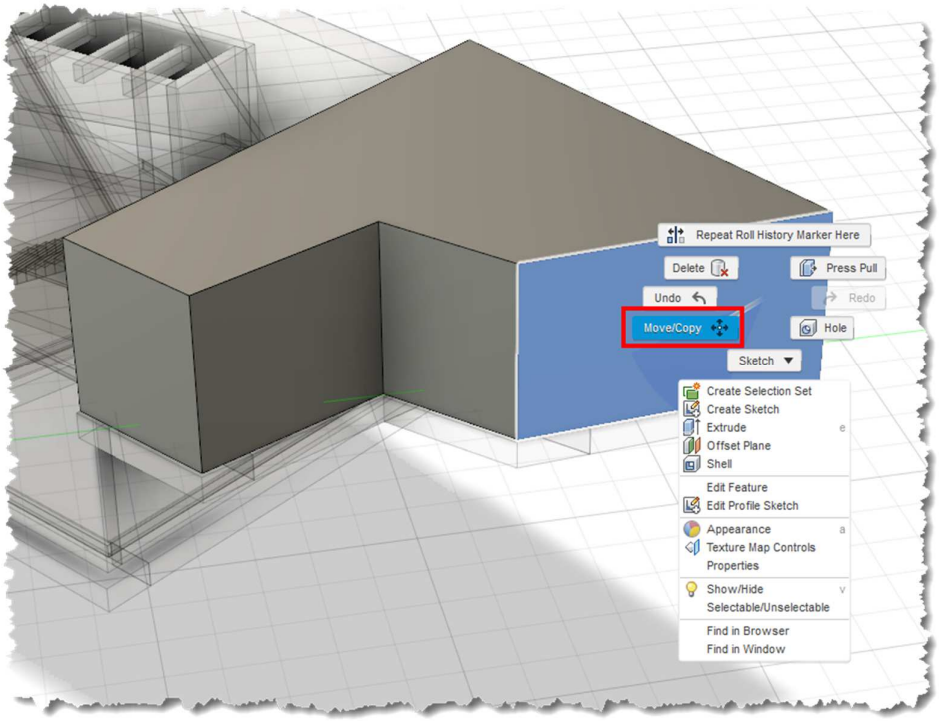
QUOTE

“The sculpture is already complete within the marble block, before I start my work. It is already there, I just have to chisel away the superfluous material”

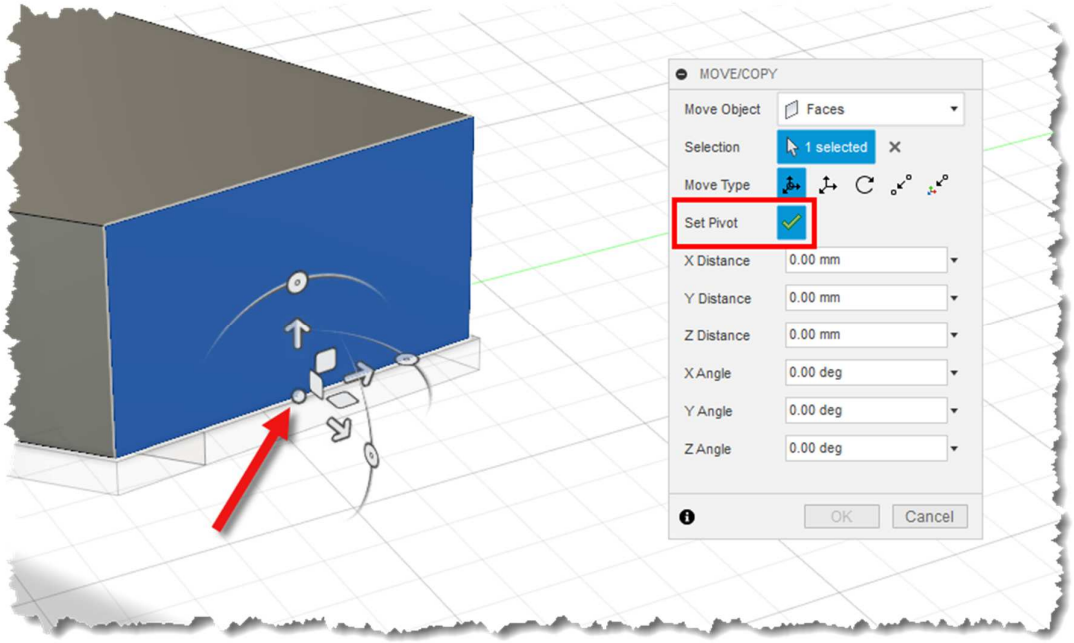
Michelangelo Buonarroti



11. Before “emptying” the interior part of the solid element, adjust the faces slope and position by using the “Move/Copy” command. Select the face close to the main entrance, right click and choose the “Move/Copy” command;

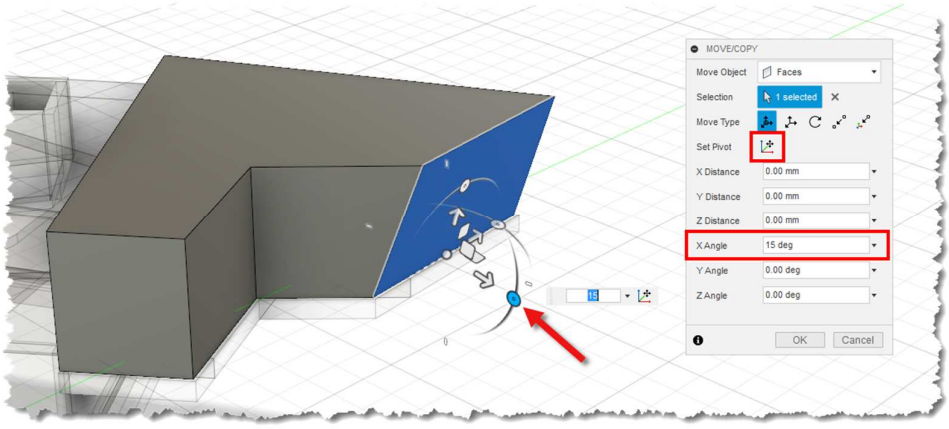


12. Click on the “Set Pivot” icon and place the pivot on the bottom line middle point; click on the green thick mark to fix the pivot position there;

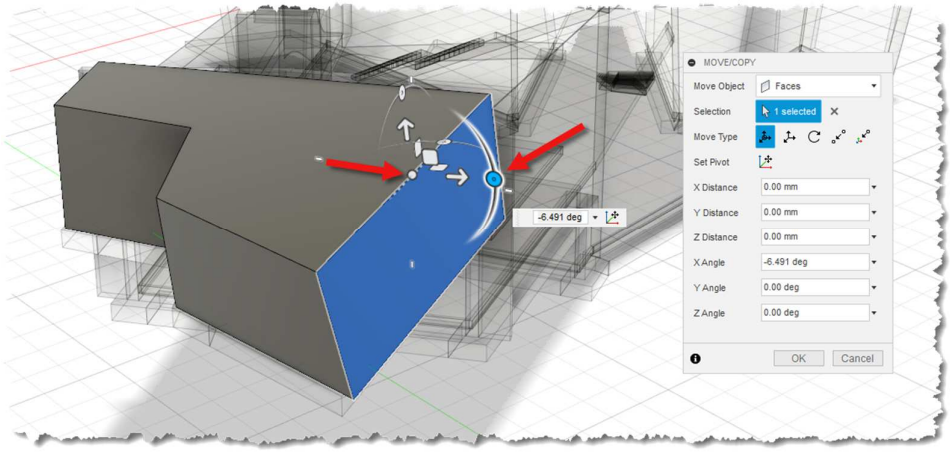


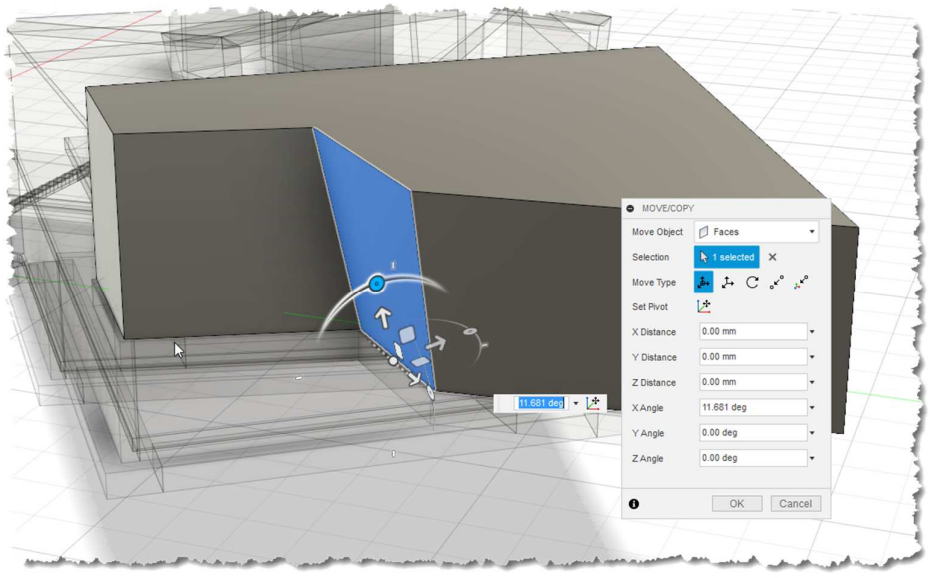
13.

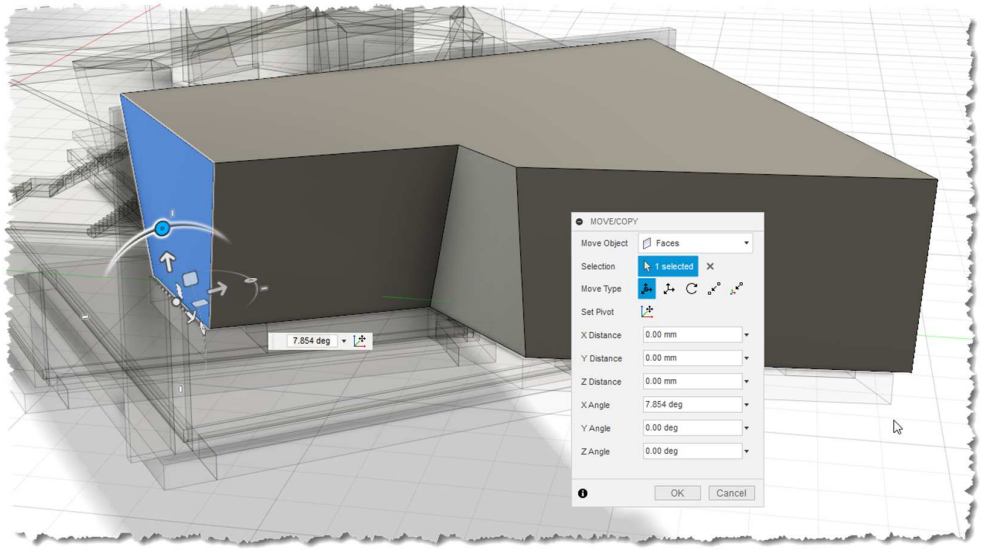
13. Notice that the “Set Pivot” icon changed when the position has been fixed. Click on the indicated rotation handle and rotate the face by 15 degrees; it is also possible to type the value on the “Move/Copy” dialog on the specific parameter;

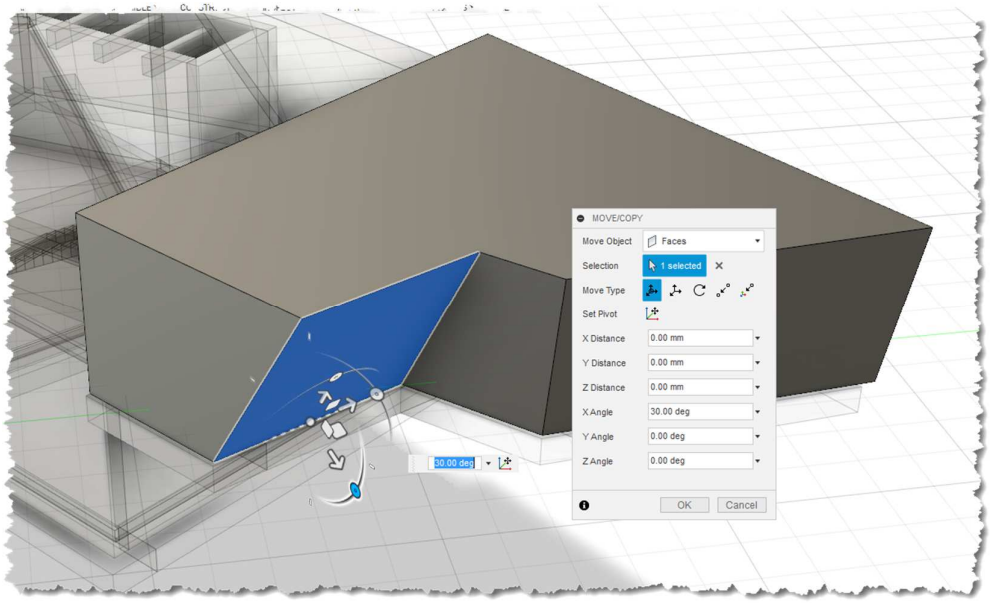


14. Rotate the other faces, too. Use the following images as a guideline;

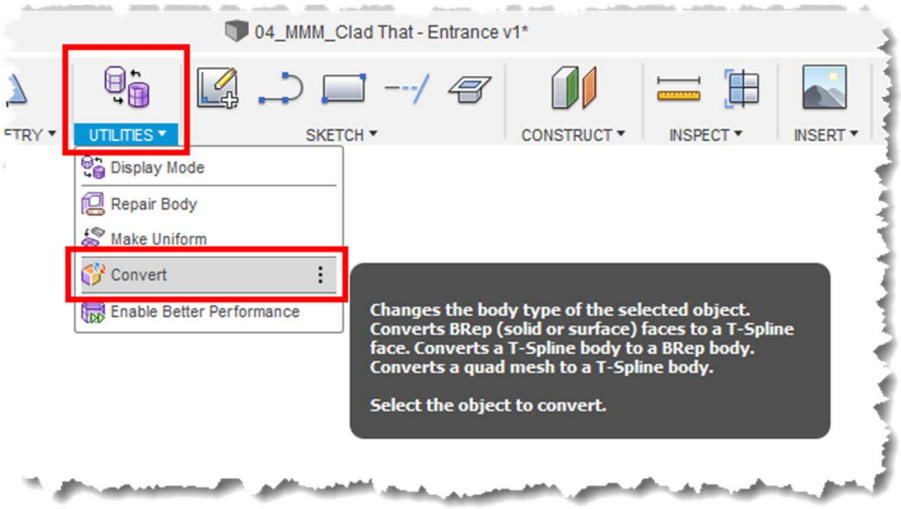




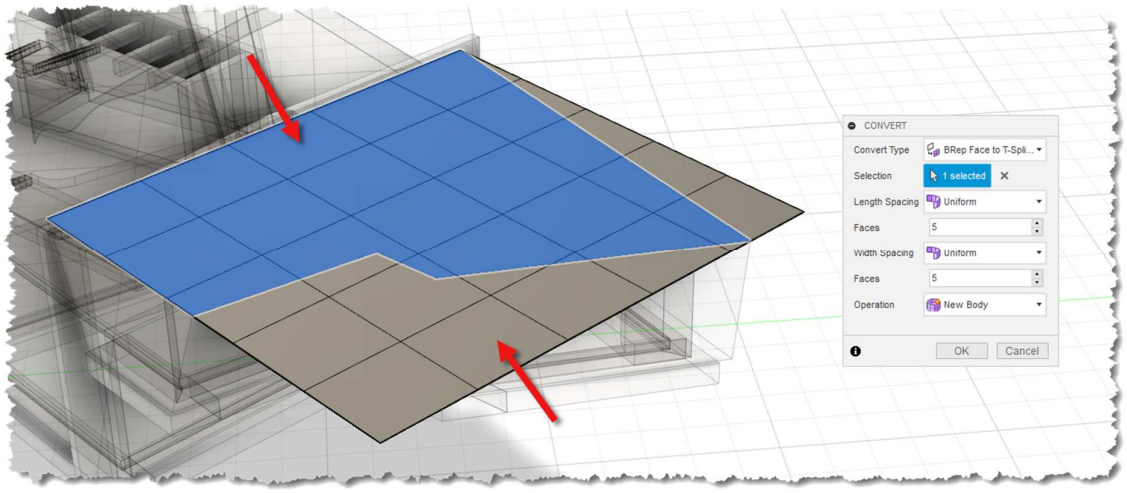

MOVE/COPY
Move Object
Faces
Selection
1 selected
Move Type
Set Pivot
X Distance
0.00 mm
Y Distance
0.00 mm
Z Distance
0.00 mm
X Angle
7.854 deg
Y Angle
0.00 deg
Z Angle
0.00 deg
OK
Cancel
7.854 deg


MOVE/COPY
Move Object
Faces
Selection
1 selected
Move Type
Set Pivot
X Distance
0.00 mm
Y Distance
0.00 mm
Z Distance
0.00 mm
X Angle
30.00 deg
Y Angle
0.00 deg
Z Angle
0.00 deg
OK
Cancel
30.00 deg

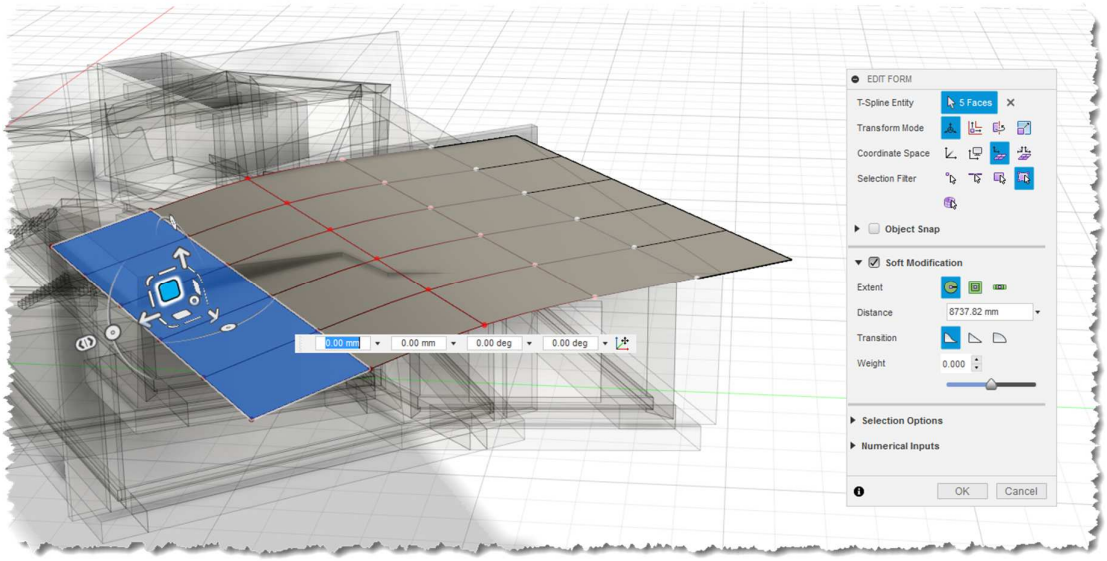
15. Enter the “Freeform” (or “Sculpt”) environment and select the “Convert” command under the “Utilities” menu;



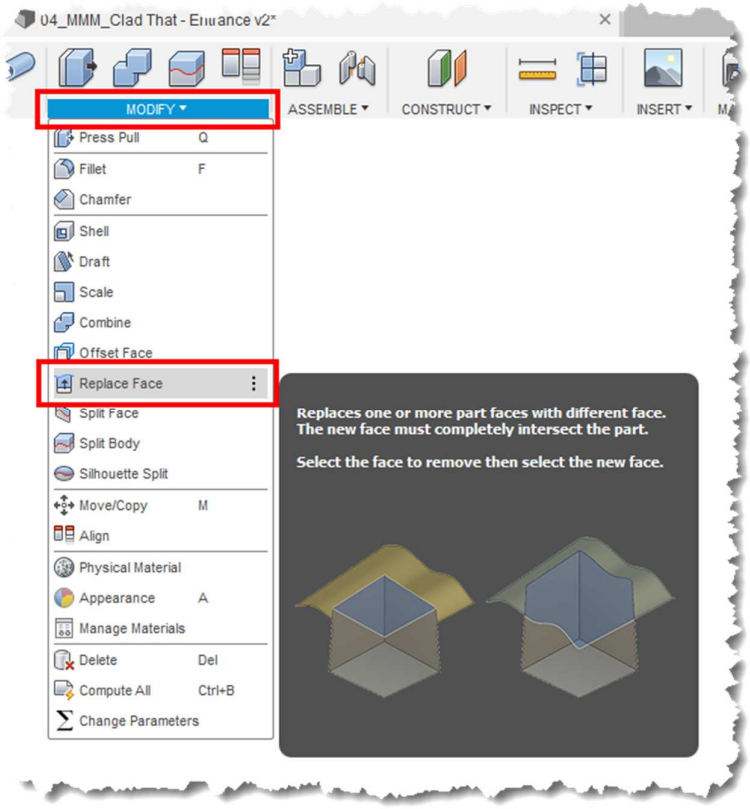
16. Specify “BRep Face to T-Spline” for the “Convert Type” setting and 5 x 5 faces for the Length and Width spacings. This command creates a T-Spline based surface using the selected one as a reference. Click “Ok” when done;



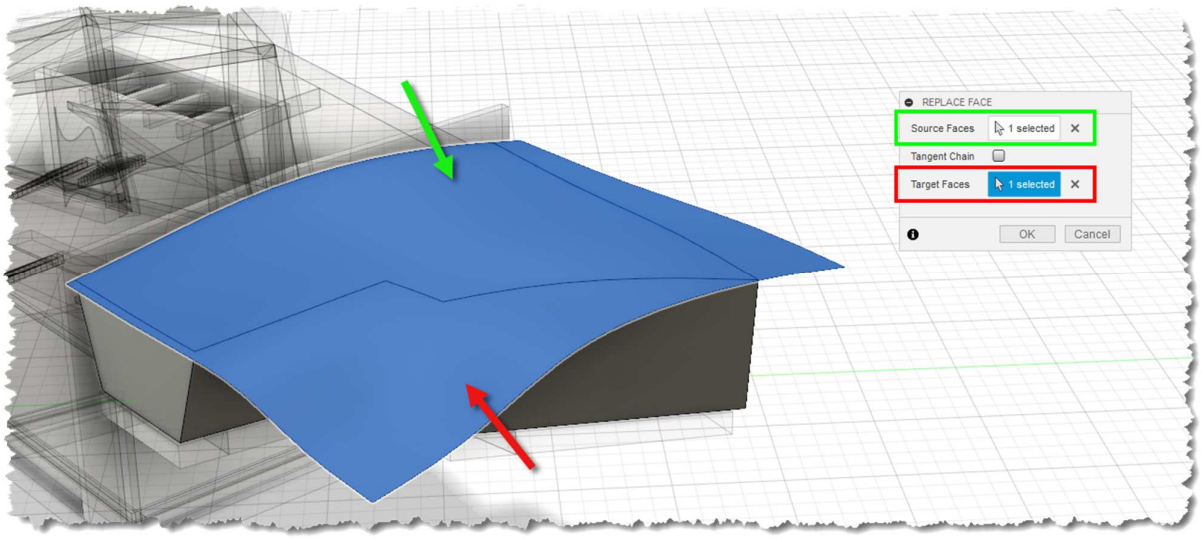
17. Edit the surface in order to create a curved canopy; use the “Soft Modification” to create a smooth transition between the modified elements and the remaining ones. End the freeform creation when done;

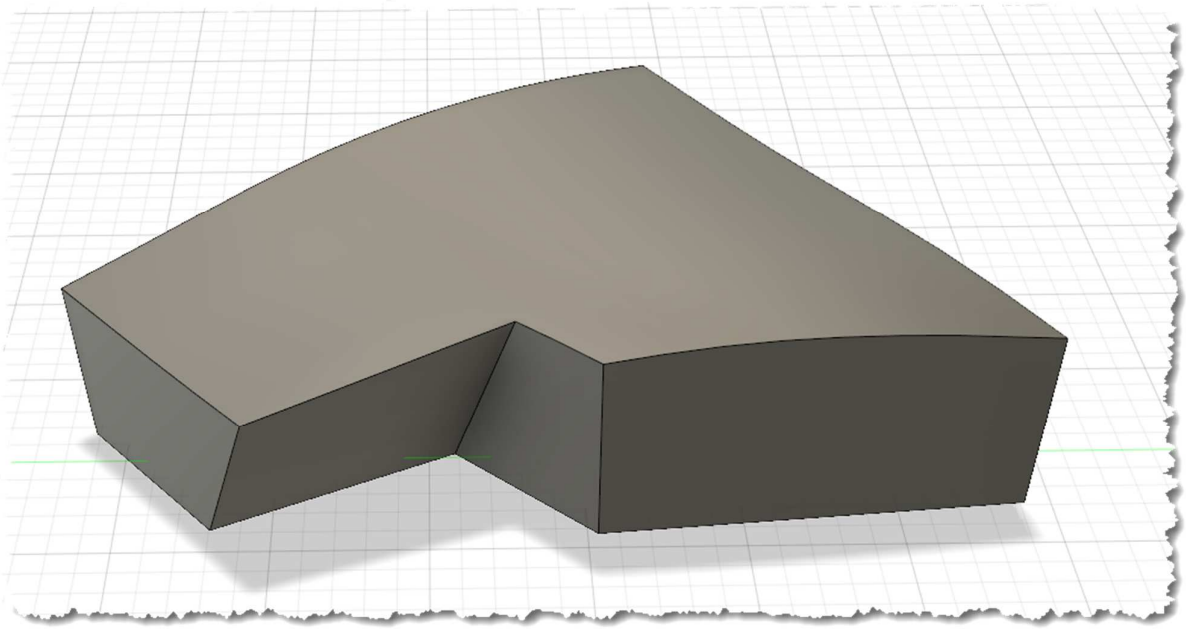


18. The scope of the next set of commands is editing the new surface to modify the top face of the existing solid geometry;
19. Click on the “Replace Face” under the “Modify” menu of the “Model” workspace;

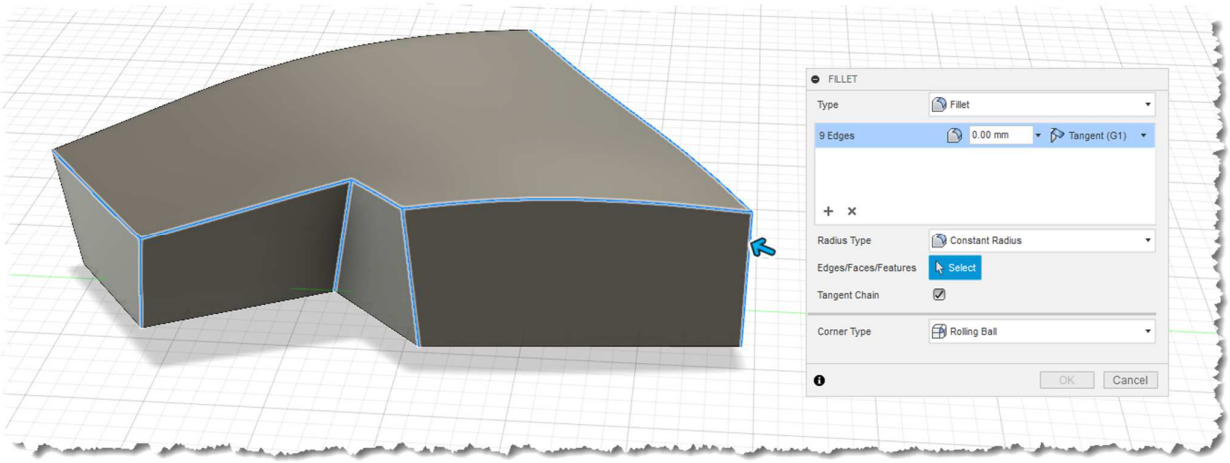


20. Select the face to be modified (Source Face) and the new surface to be used to replace the original face (Target Face); Click “Ok” when done;

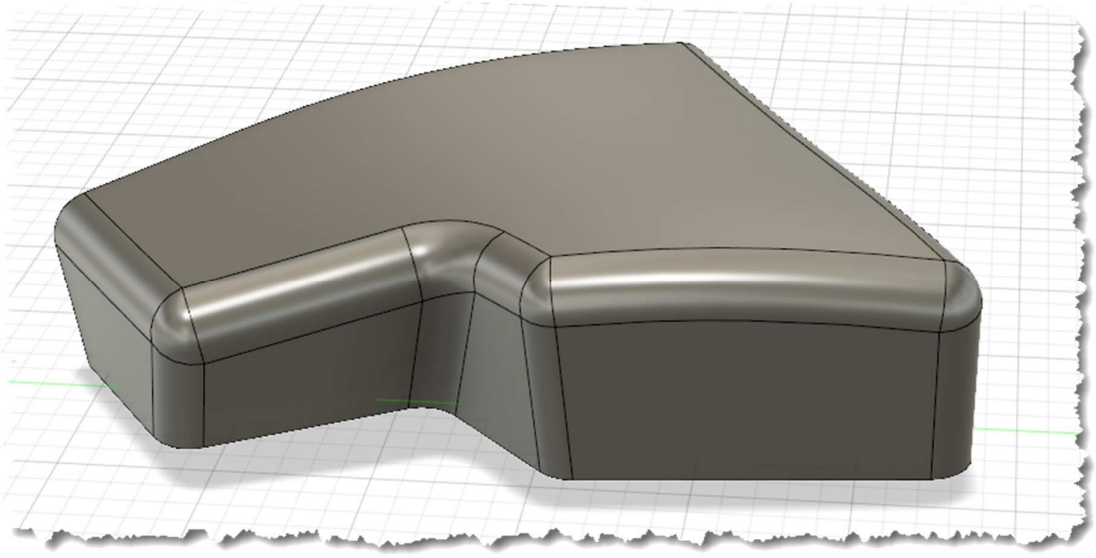




21. Once the face has been replaced, use the “Fillet” command to create curved transitions on the highlighted corners. Specify 700 mm as the fillet radius;



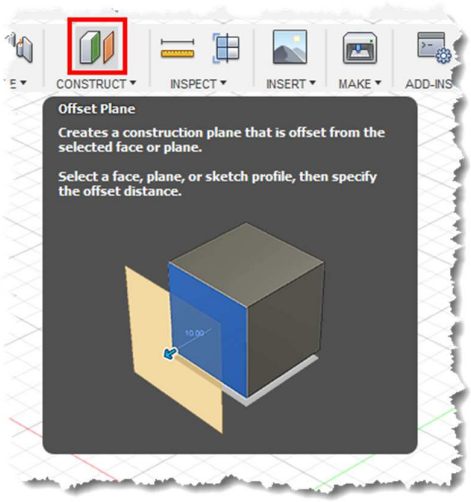
22. The result should be like that one on the next image;



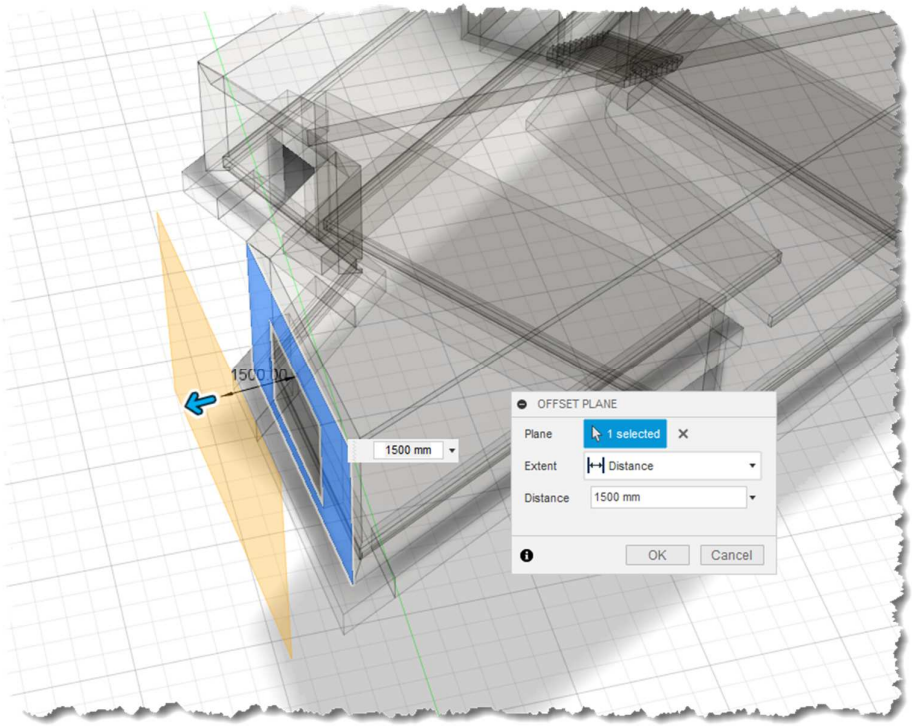
12.2 Opening creation

The scope of this exercise is showing how to create the two openings on the front façade and to connect them to the main volume. The openings will be defined using two solids; these are based on the openings defined in the structural model and will take those as a reference.

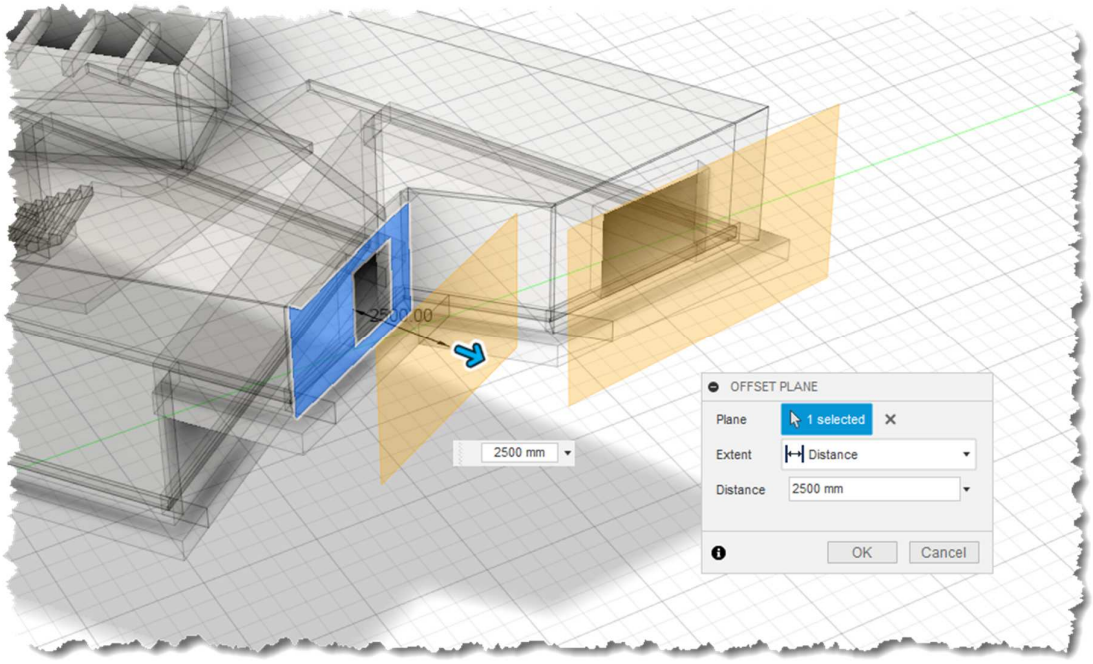
1. Turn on the visibility of the structural model;
2. Click on the command “Offset Plane” to create a new plane based on an existing one;



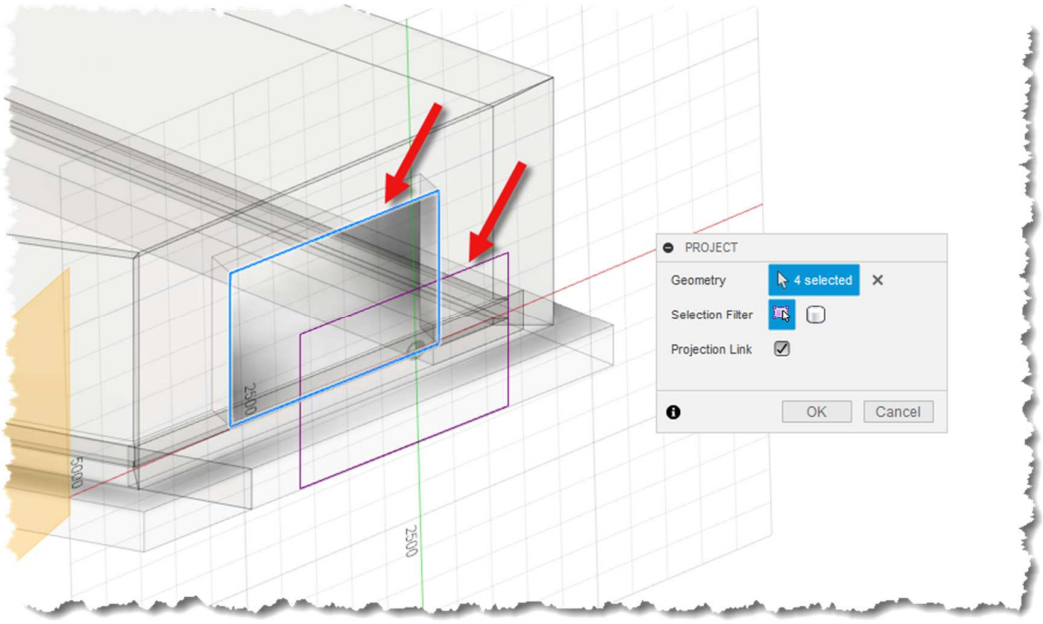
3. Select the entrance face to indicate the plane reference and specify 1500 mm as the “Distance”;



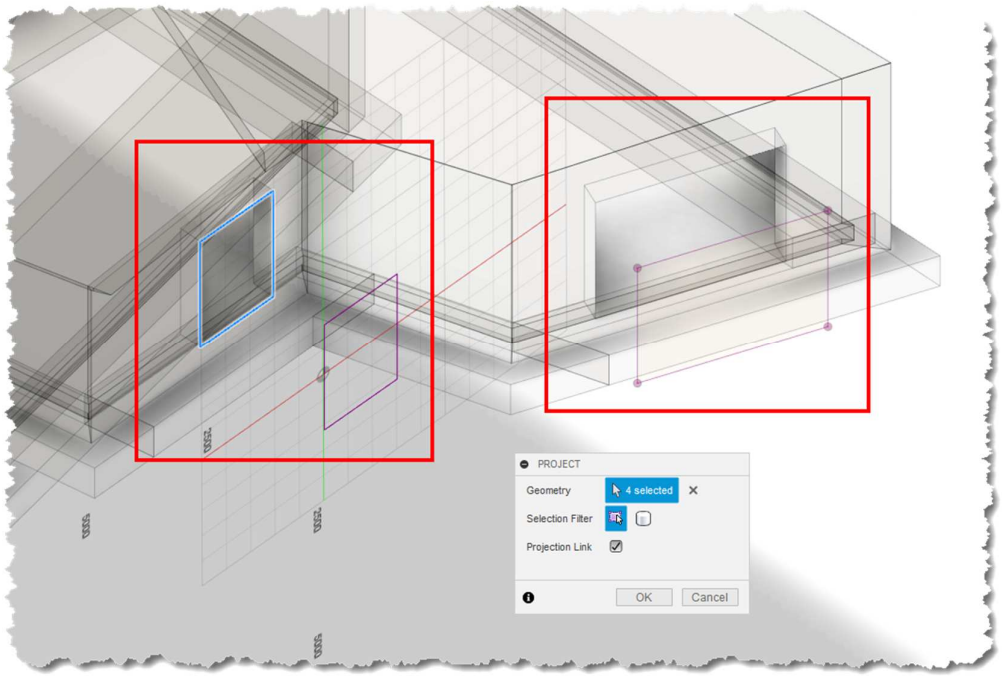
4. Repeat the same on the other face as indicated and set 2500 mm as offset distance;



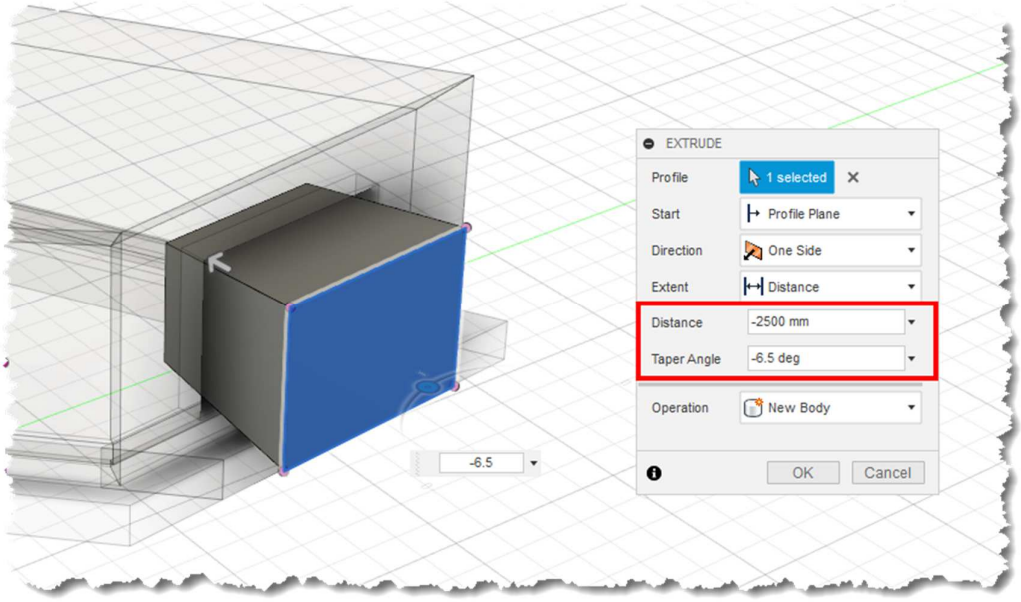
5. Start a new sketch using the first plane as the host and use the “Project” command to capture the door’s edges;



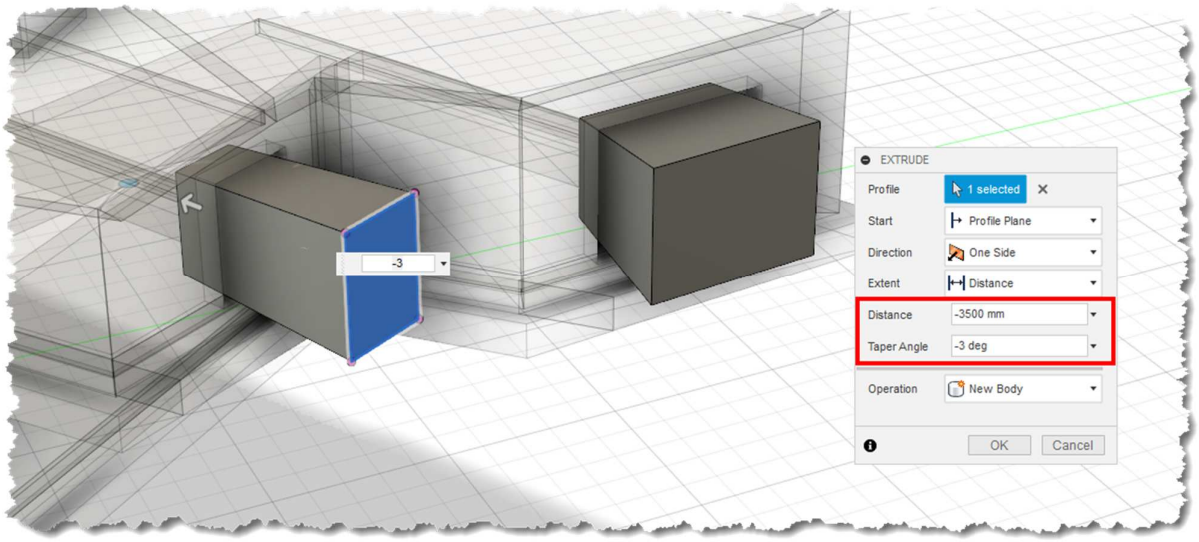
6. Repeat the same operation creating another sketch and projecting the lines of the second door;



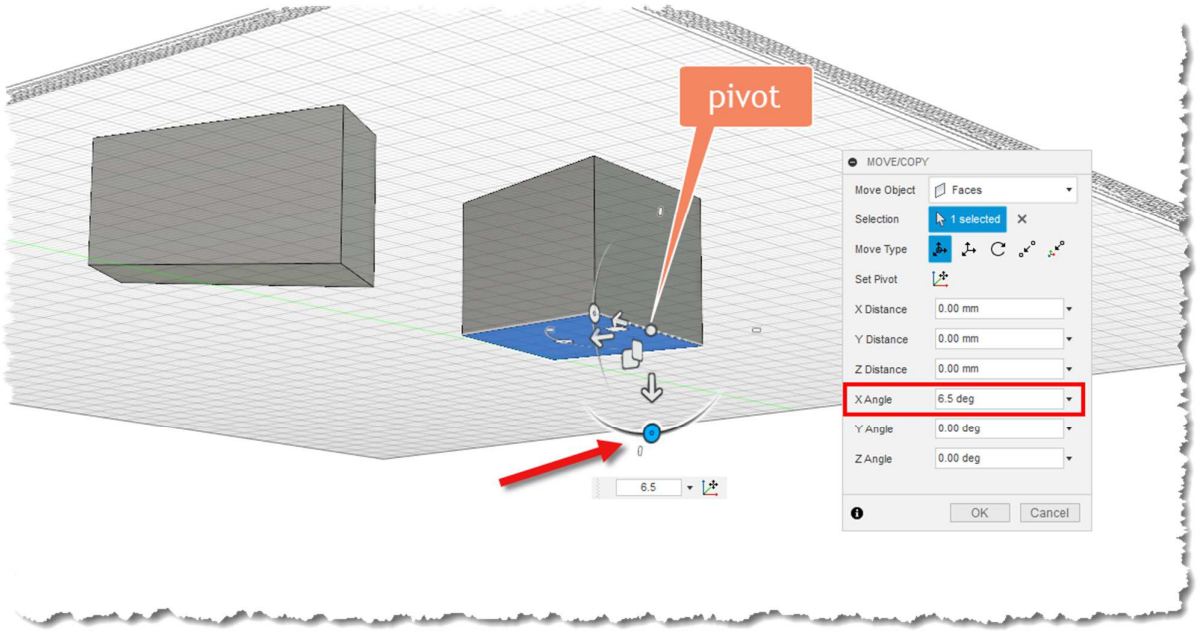
7. Use the extrude command to create a solid out of the first sketch. Notice that the extrusion definition can be driven by its distance and by a taper angle. Set -6.5 degrees as taper angle and a distance of -2500 mm (the "minus" sign depends on the host plane orientation, in some cases it could be necessary to set a positive distance instead). Click "Ok" when finished;



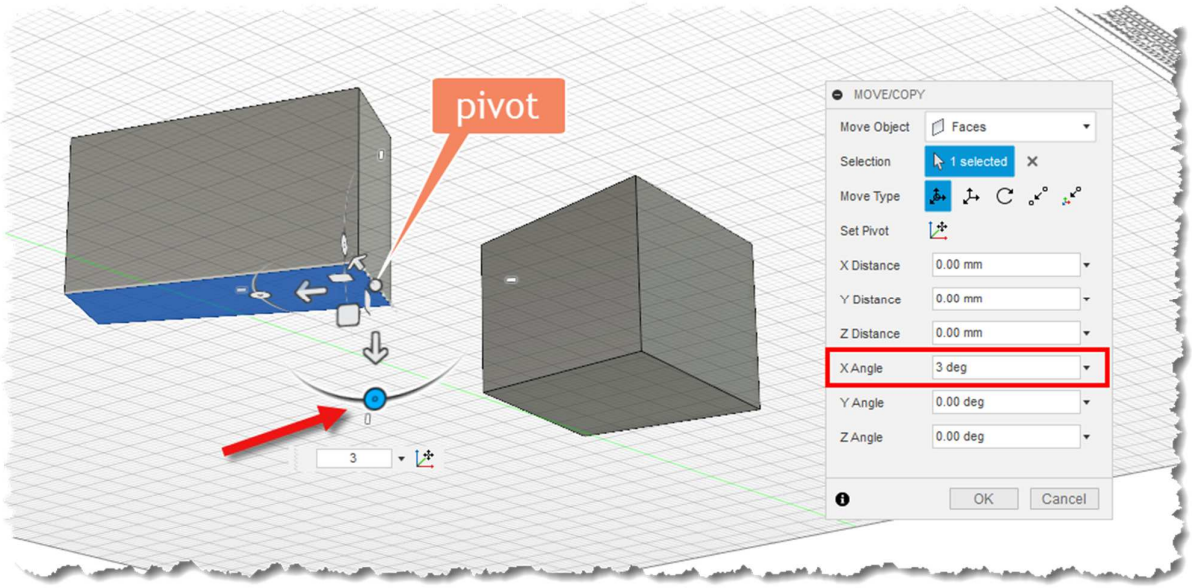
8. Do the same for the second sketch, setting this time -3 degrees for the angle and -3500 mm for the distance. Click “Ok” when done;



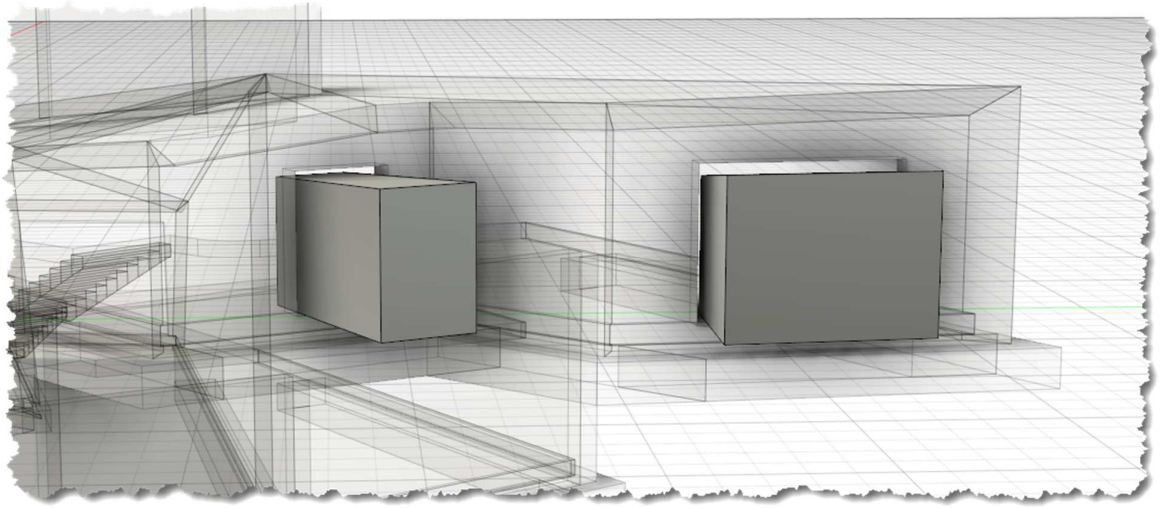
9. The two solids have a slanted bottom face; these elements must be horizontal to match the floor face orientation;
10. Select the face, right click and chose the “Move/Copy” command; set the Pivot in the indicated point and the specify 6.5 degrees for the X angle value. This rotates the face and makes it to be horizontal;



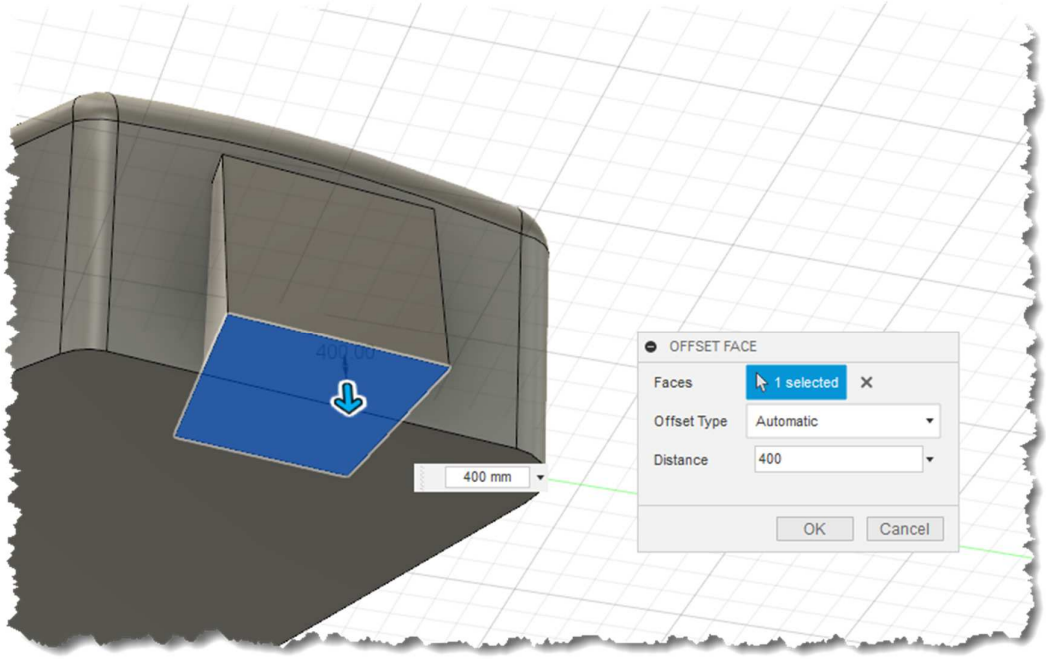
11. Apply the same technique to the other bottom face and specify an angle of 3 degrees this time;

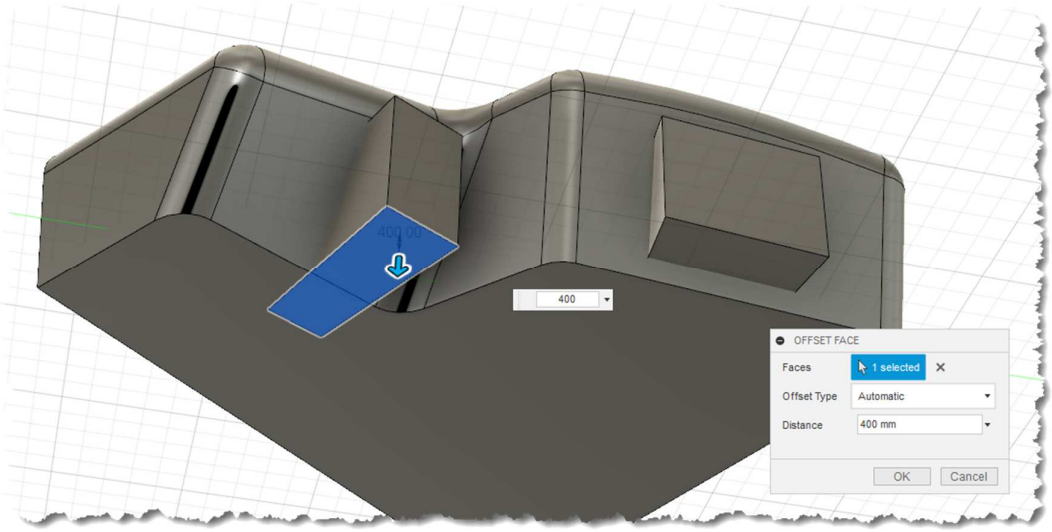


12. The results should be like the next image;

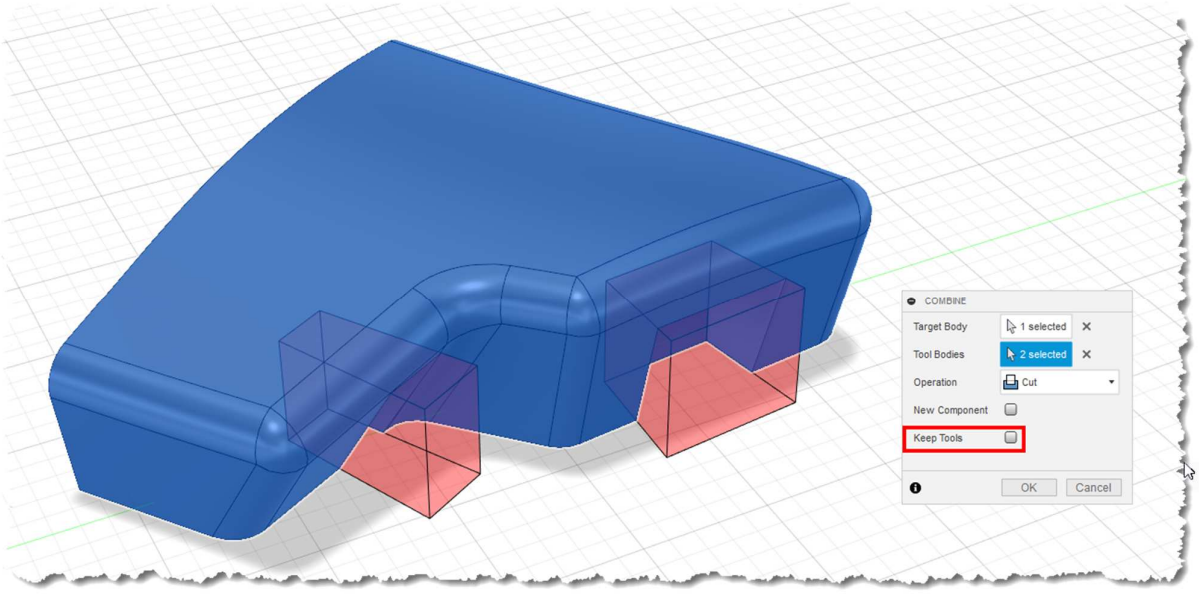


13. Using the “Move/Copy” command, move both the faces 400 mm downwards to match the main solid bottom face elevation;

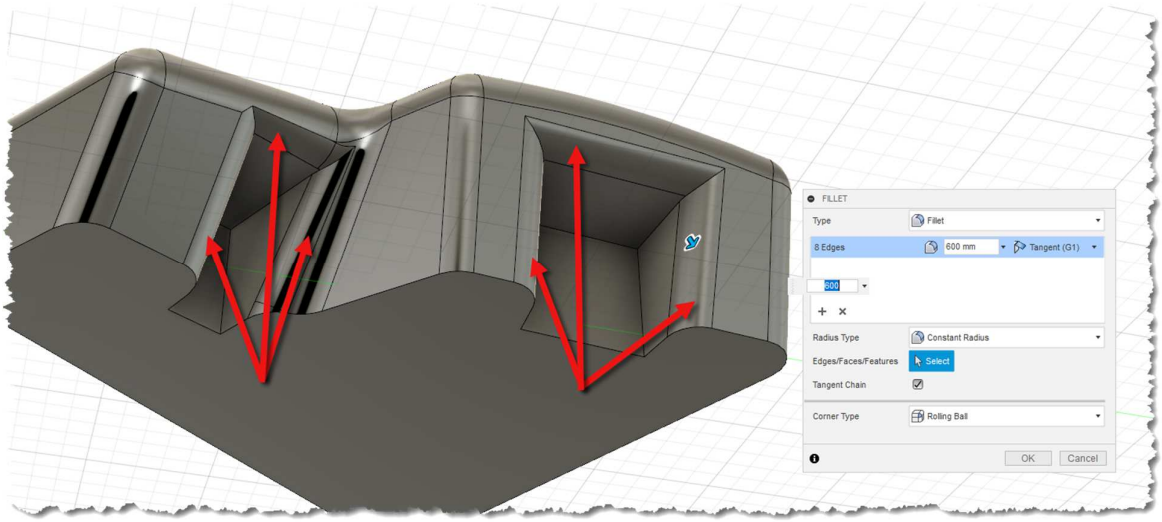




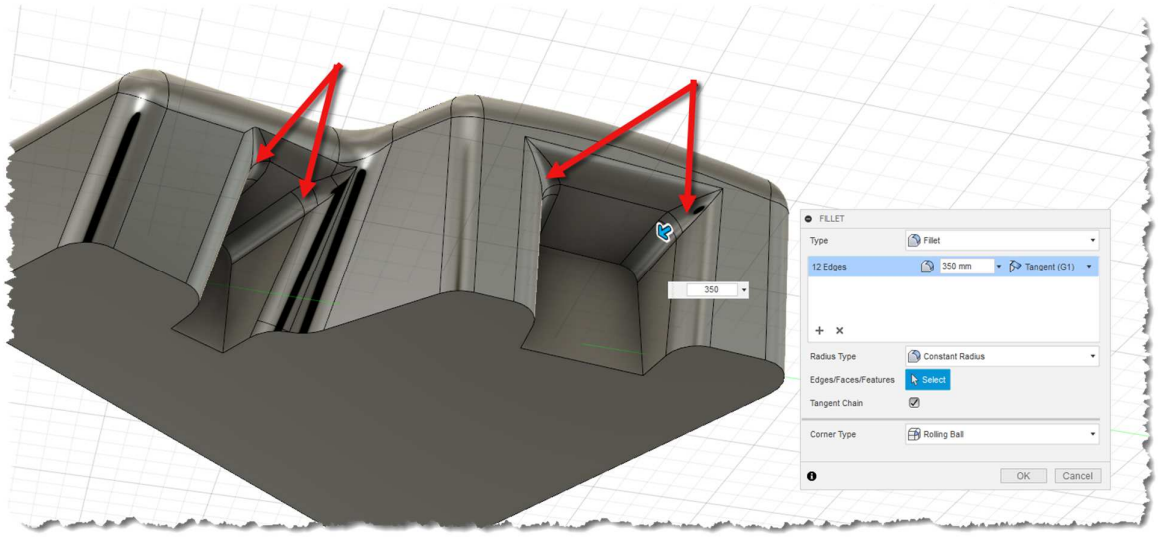
14. Use the “Combine” command to cut the main solid with the two elements modeled for the openings. The option “Keep Tools” is not needed here (the cuts volumes are not going to be reused anymore), so uncheck it;



15. Use the “Fillet” command to create round edges as indicated below. Specify 600 mm as the radius value;



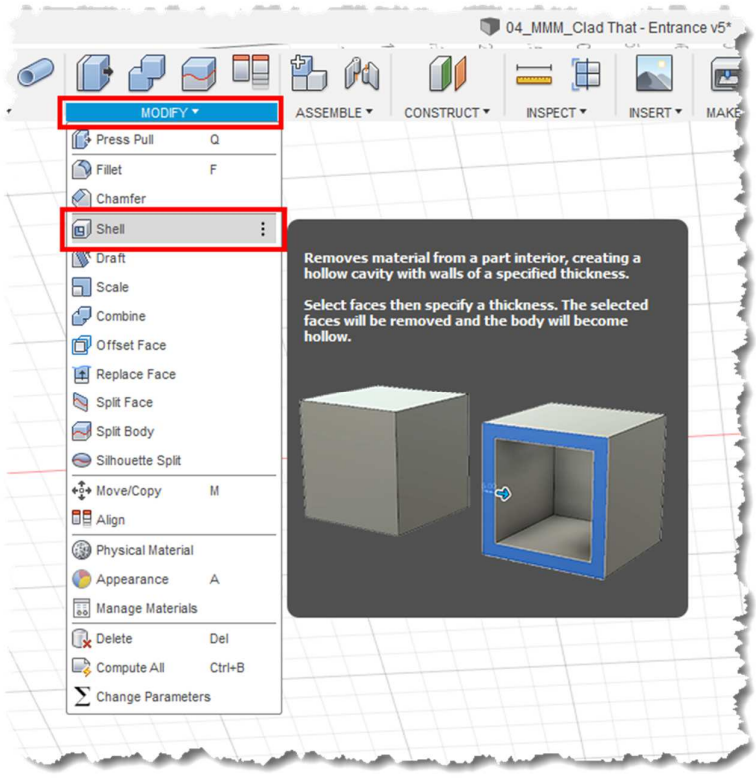
15. Repeat the same command to fillet the interior edges with a radius of 350 mm;



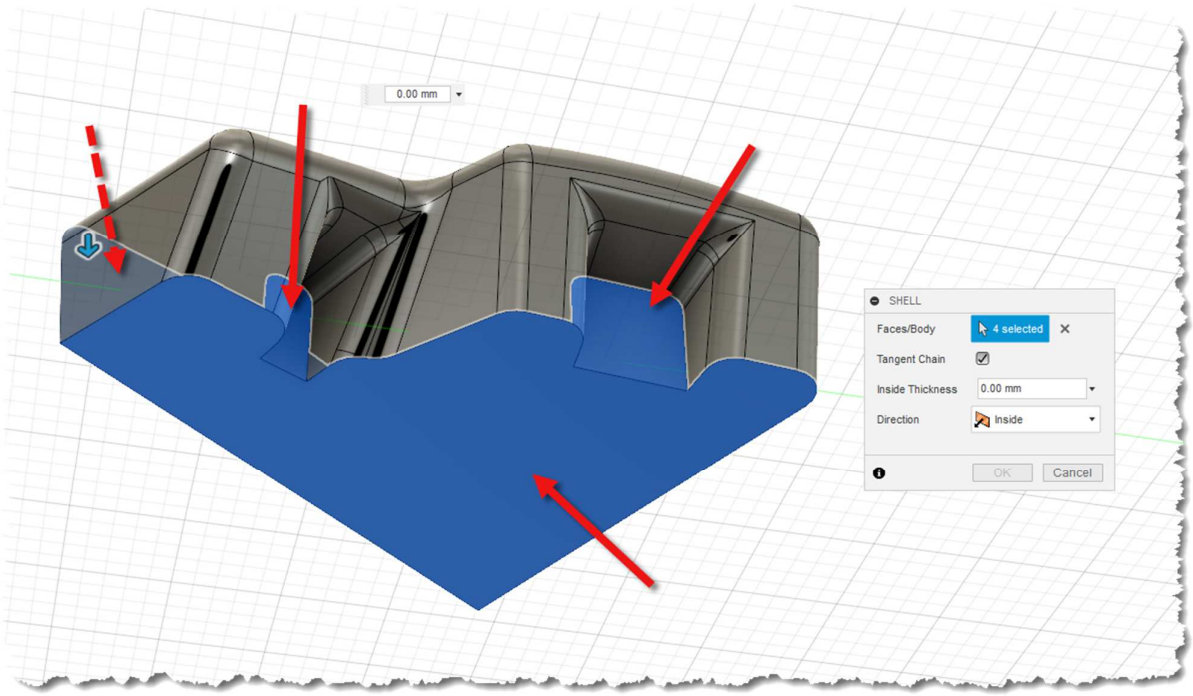
12.3 Body Shell

The created solid must be emptied, and the resulting thickness must be the same of the rest of cladding elements.

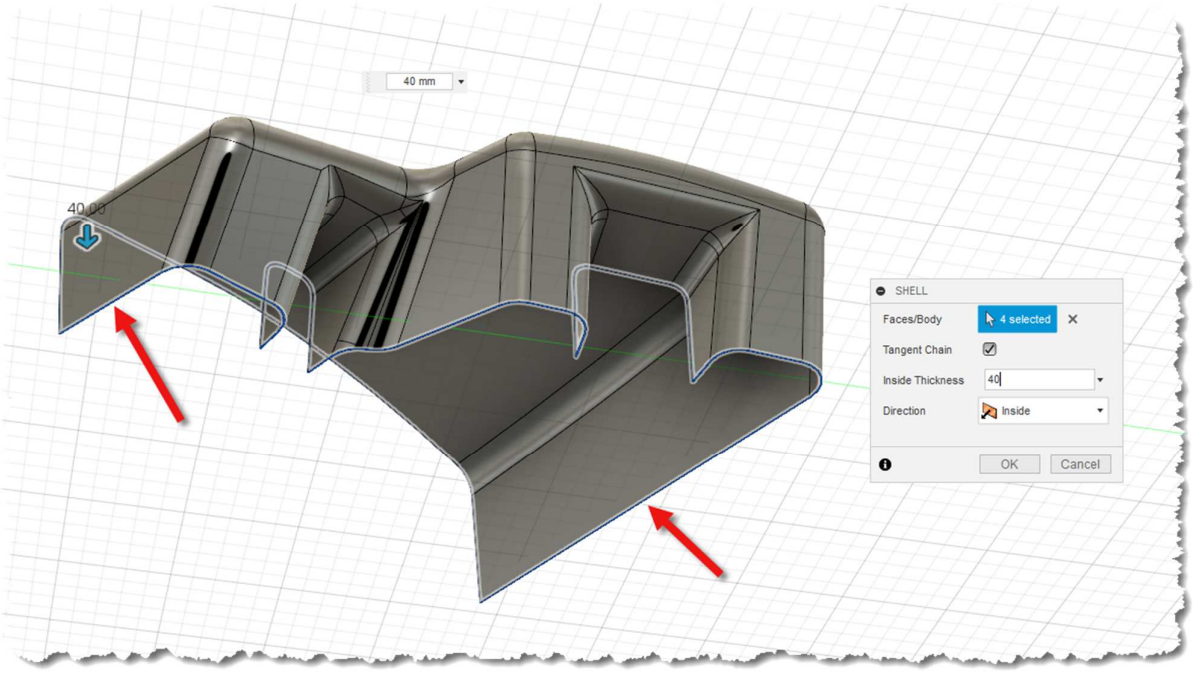
1. Click on the “Shell” command, under the “Modify” menu of the “Model” workspace;



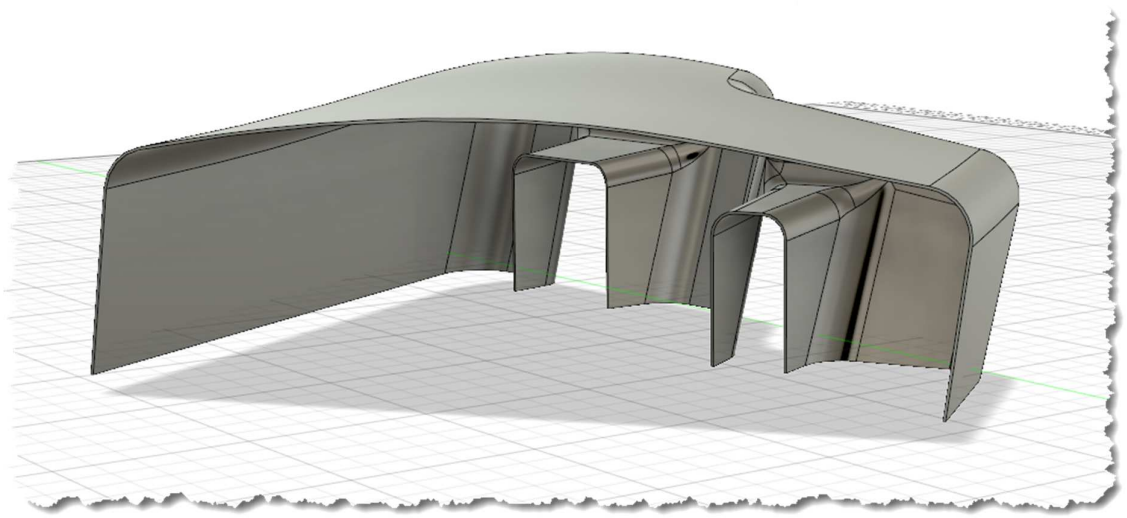
2. Select the faces to be used for driving the shell feature. These faces indicate the various directions in which the solid shelling will be proceeding;



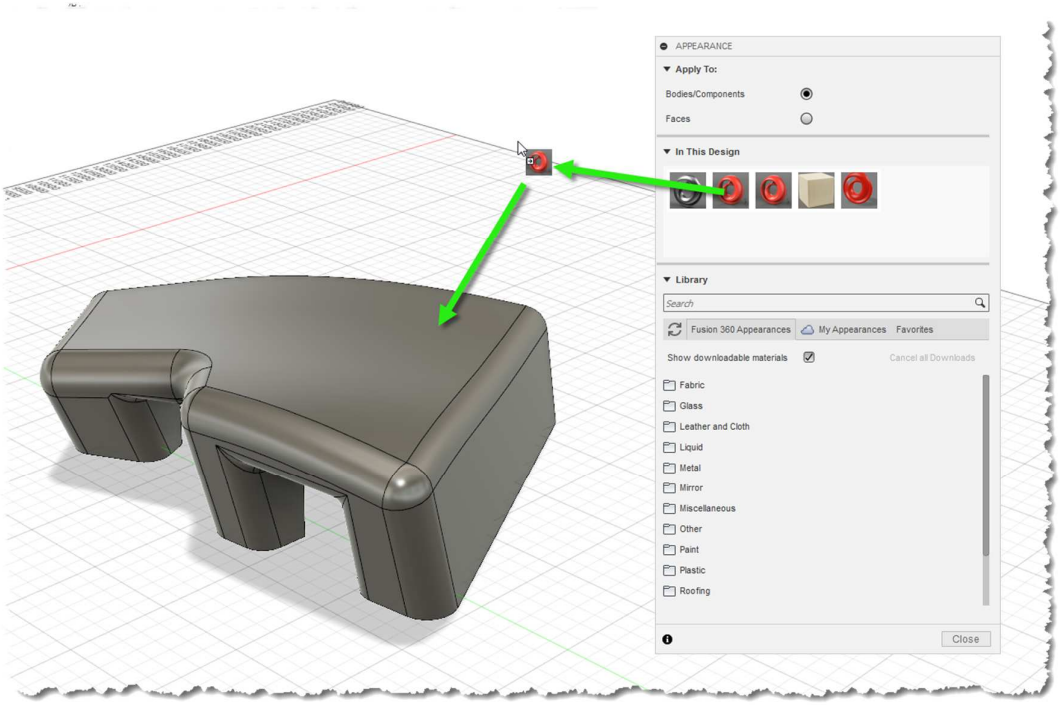
3. Specify 40 mm as the “Inside Thickness” value. Before clicking “Ok”, make sure that the shell operation is applied on the right direction (inwards in this case);



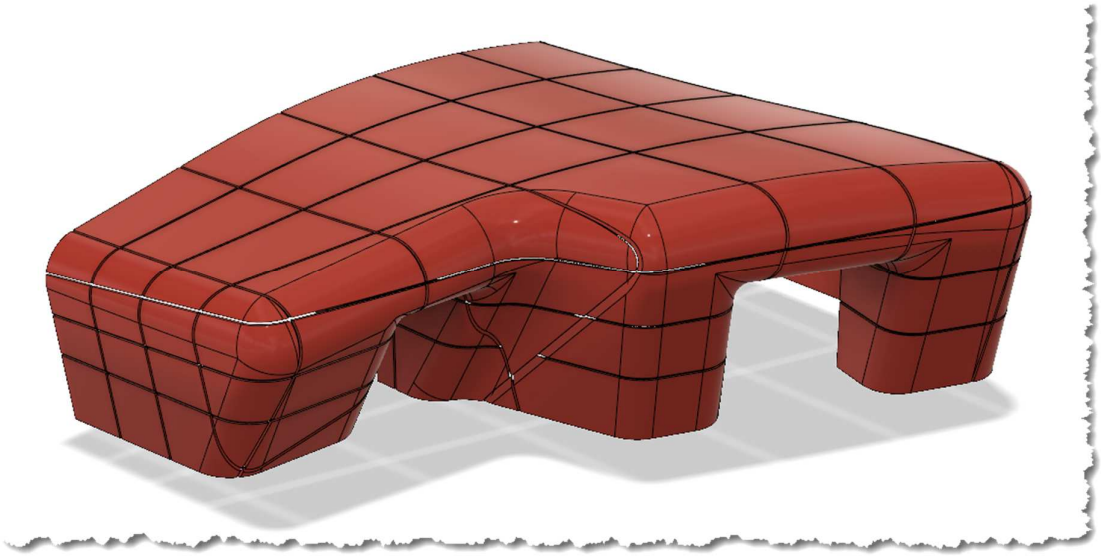
4. This command will produce an element similar to that one on the following picture;



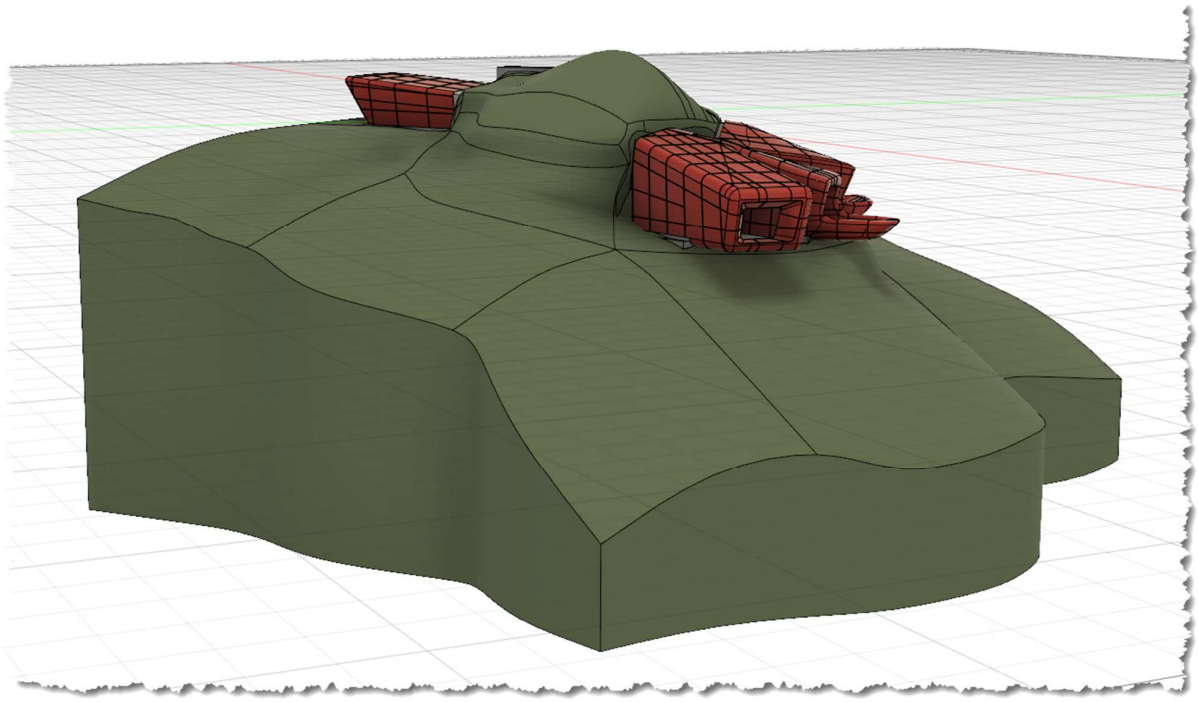
5. Apply a material if desired;



6. Proceed with making the cuts to divide the continuous shape in panels as shown in a previous chapter of this document. This procedure requires to:
    a. Create sketches
    b. Extrude the sketches using the Patch environment;
    c. Thicken the extruded surfaces;
    d. Use the Combine command to cut the shell in panels
    e. Grouping the cut elements is a good idea to maintain the model under control and avoid having too many body elements under the “Body” group of the project tree.



13 Exercise #5: Terrain Creation

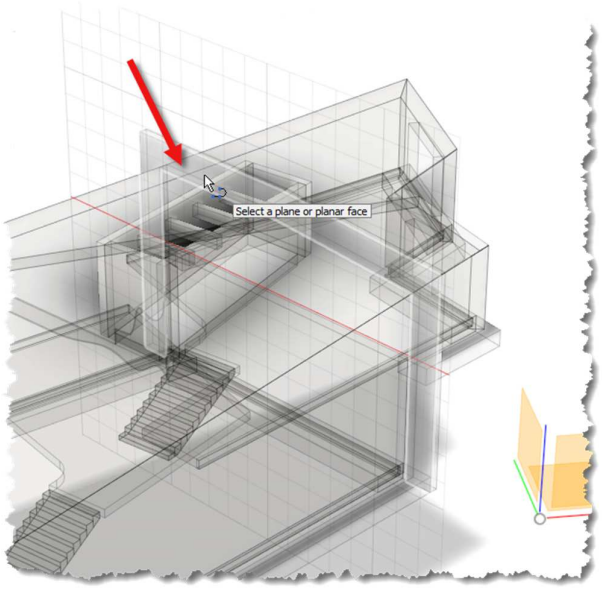


The scope of this exercise is showing methods and techniques to model the terrain shape; this will be excavated by the building so that the model will have a sort of tunnel in it. This is not possible to be achieved in Revit so Fusion 360 is used to define the terrain geometry and later this will be imported in Revit to define the toposurface.

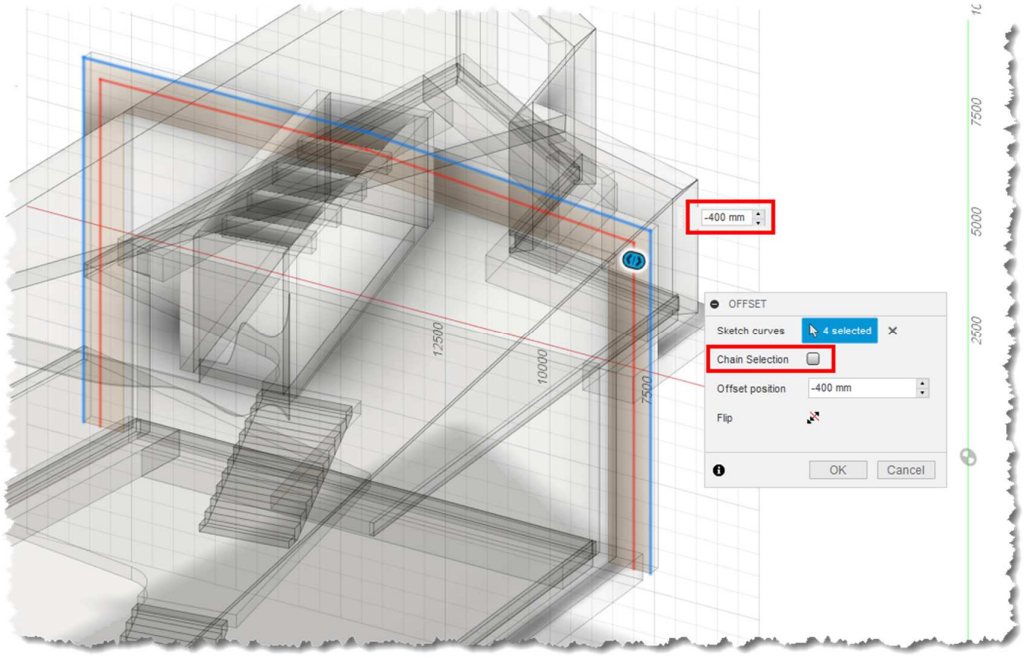
Use the “05_MMM_Clad That - Terrain.F3D” to create a new design or open it from the A360 cloud project.

13.1 Digital Terrain Model setup

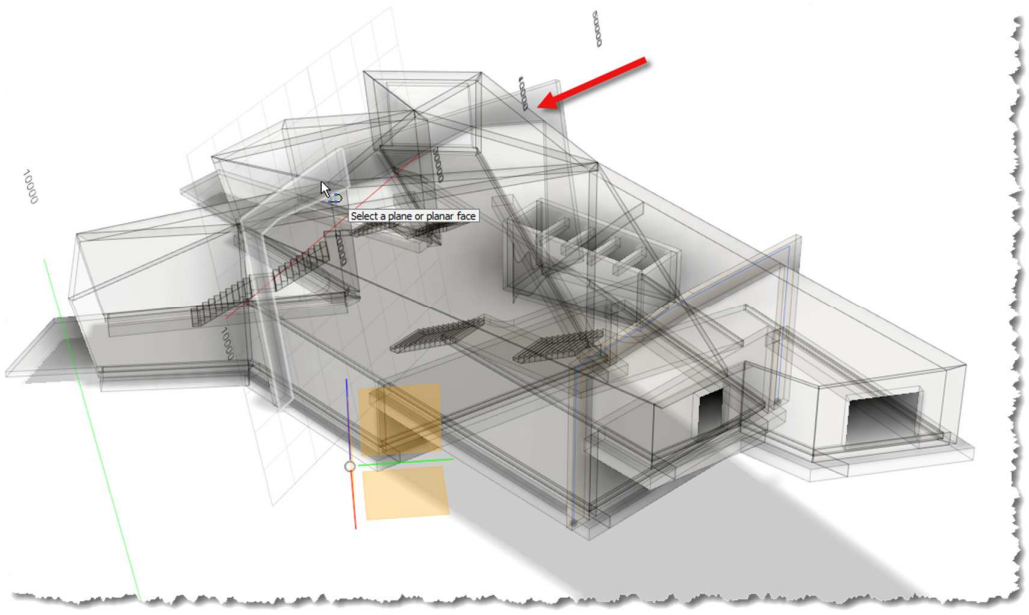
1. Create and activate a new component and call it “06_Terrain”;
2. Leave visible the “01_MMM_Structure” and the “03_Boundaries” components only;
3. The terrain will be shaped as a “hill” between the two retaining walls; this will be connected with the foundations below the cladded building branches;
4. Create a sketch using the indicated plane on the top retaining wall;

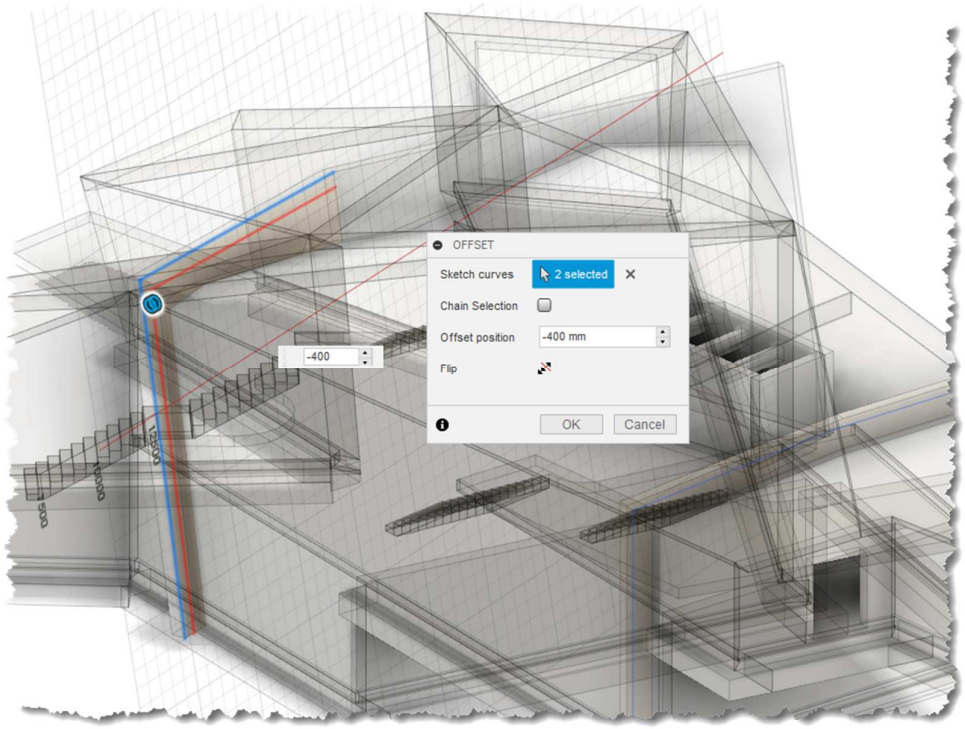


5. Make an offset of the outer profile; set 400 mm as the distance (notice that the distance could be either positive or negative depending on the orientation of the host plane);

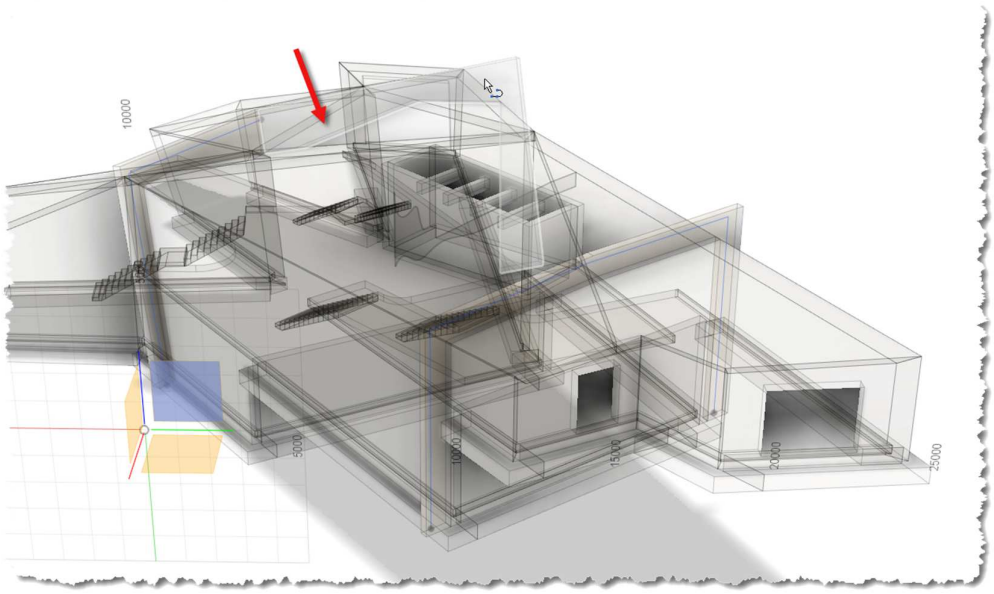


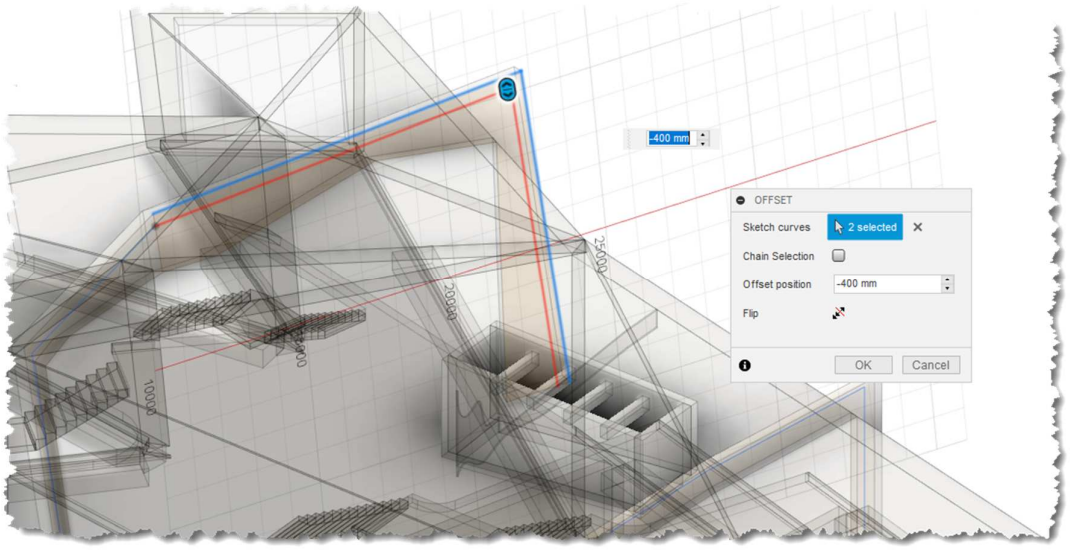
6. Do the same with the indicated faces of the lower retaining wall;



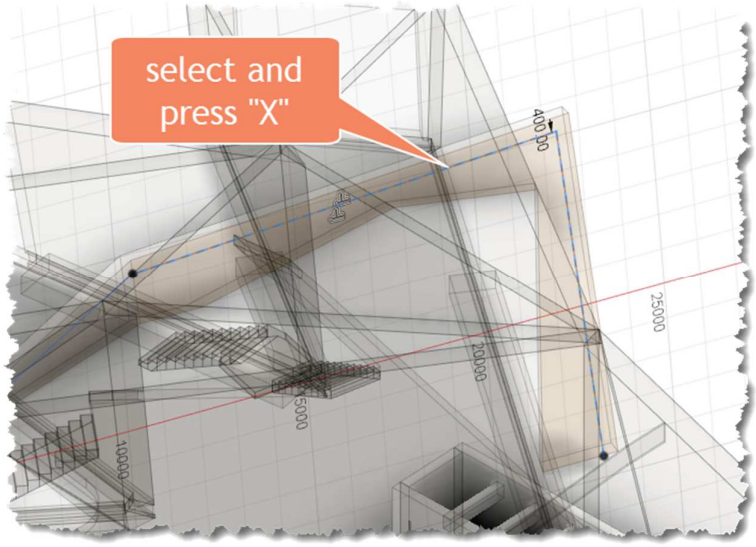


7. Being that the retaining wall is not planar, repeat the same process for the part on the right; after the offset is done, do not finish the sketch;

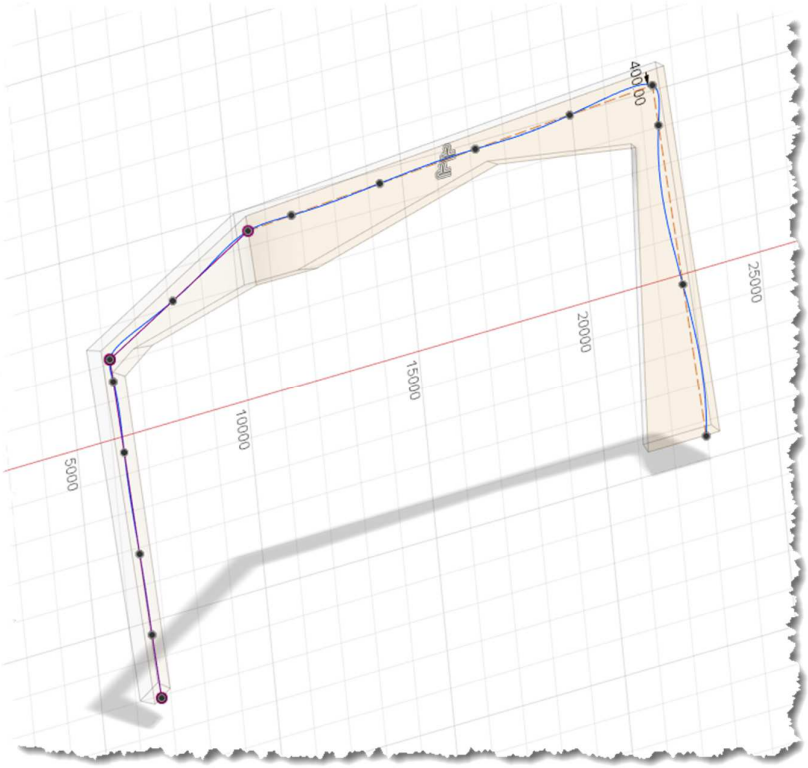




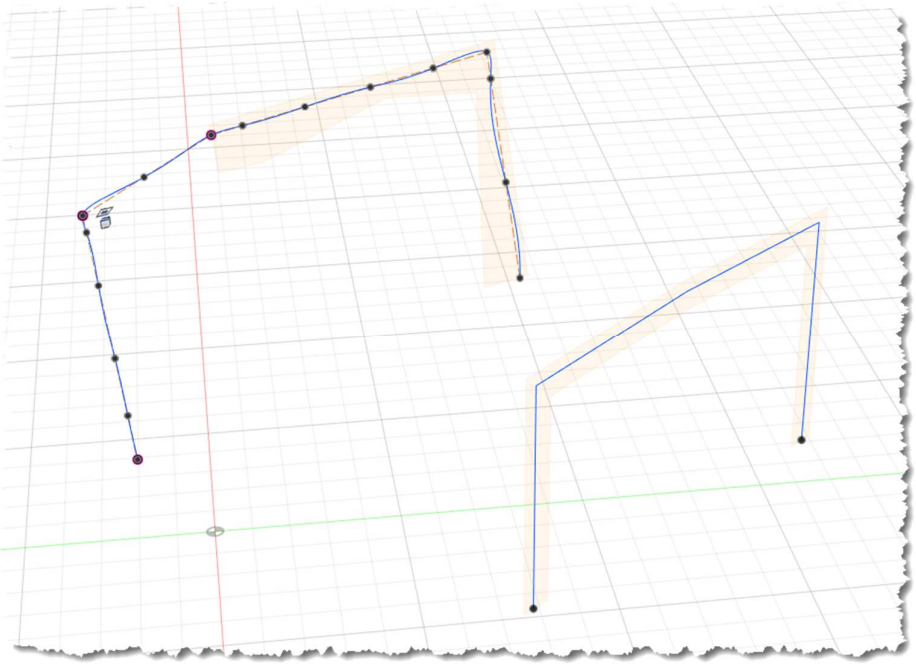
8. Select the offset lines and press “X” to transform them to construction lines inside the sketch. They will then appear dashed;

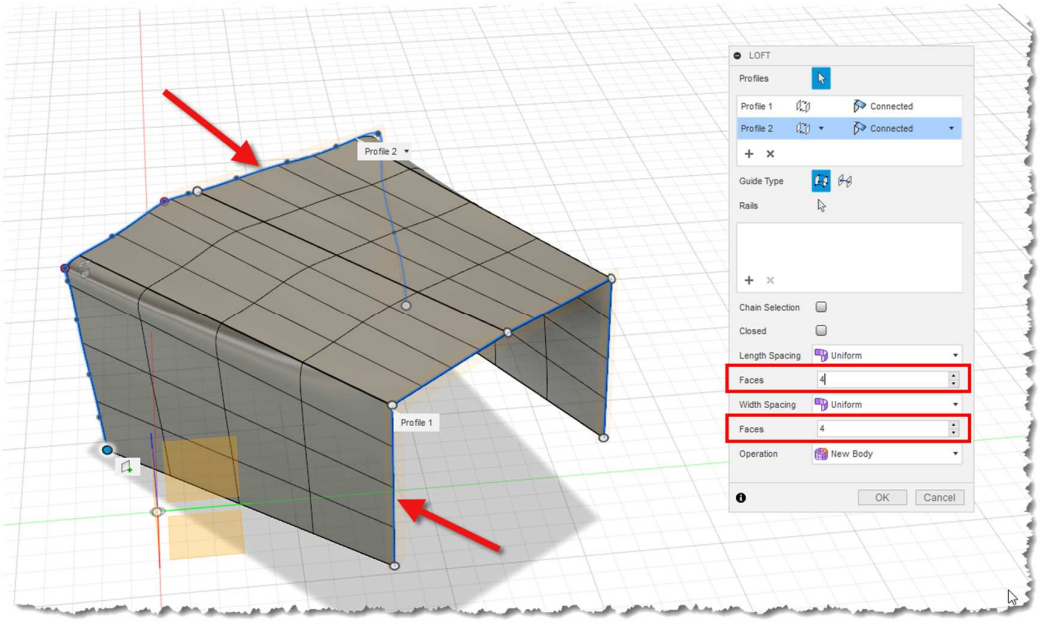


9. Create a “Spline Through Fit Points” to approximate the two profiles on the lower retaining wall with a continuous curve;

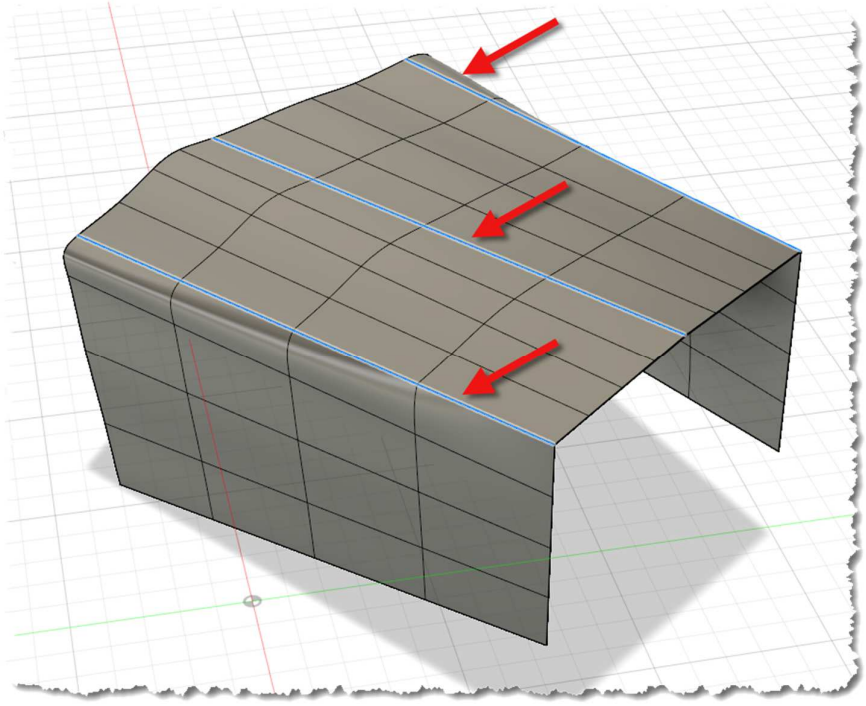


10. Enter the Freeform environment to create a loft between the profiles related to the top and lower walls;

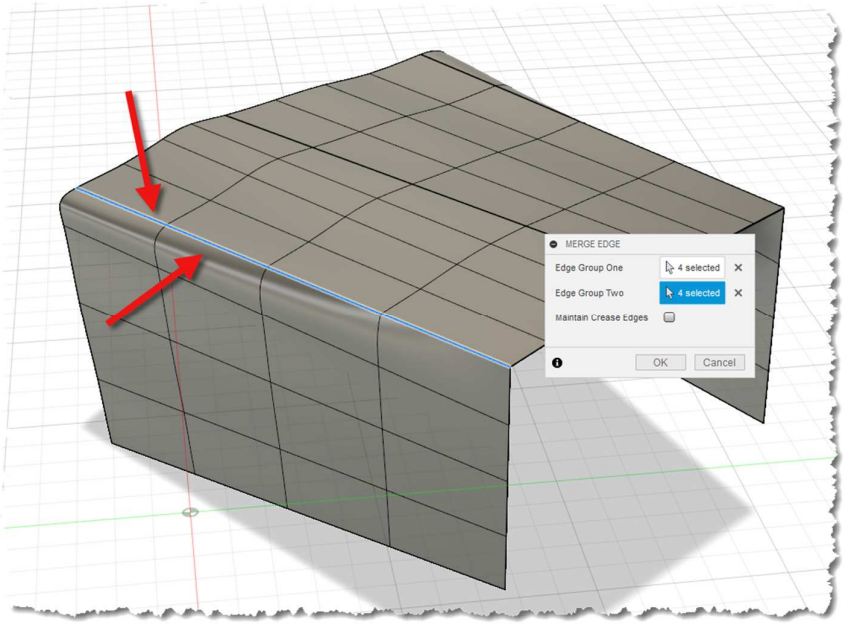




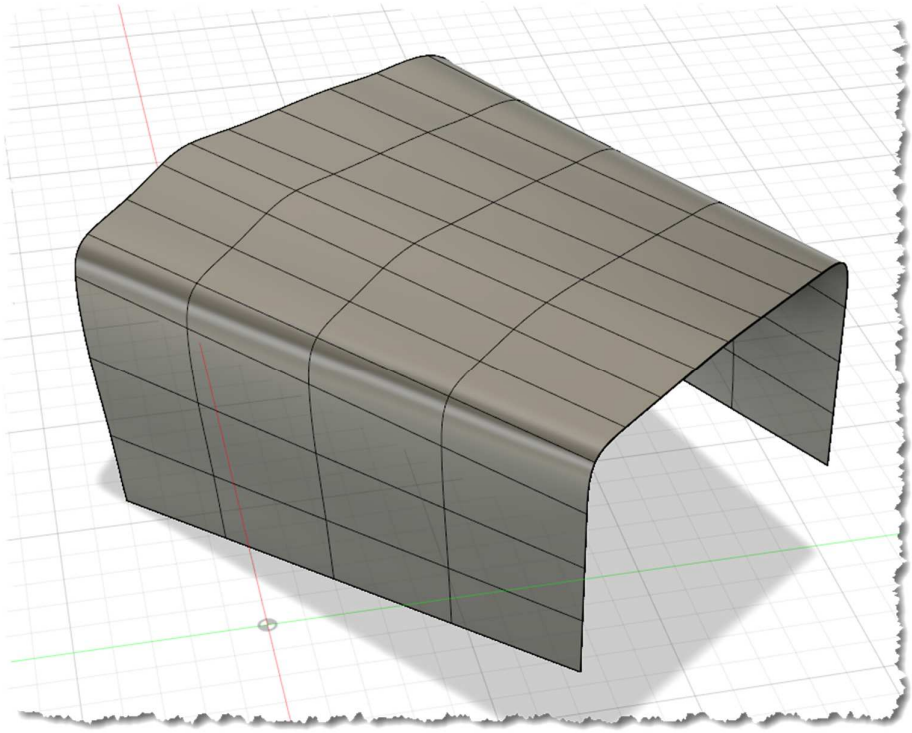
11. The vertices on the first profile make the surface to be composed of several sub-surfaces; these need to be connected using the “Merge Edges” command;



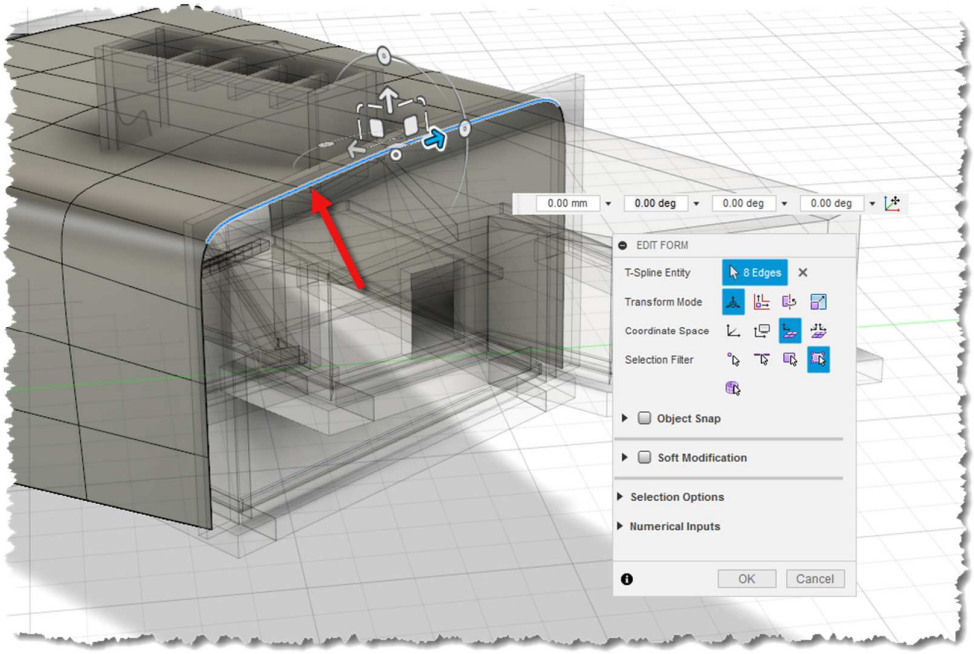
12. Repeat the “Merge Edge” command for each set of lines indicated in the previous image;



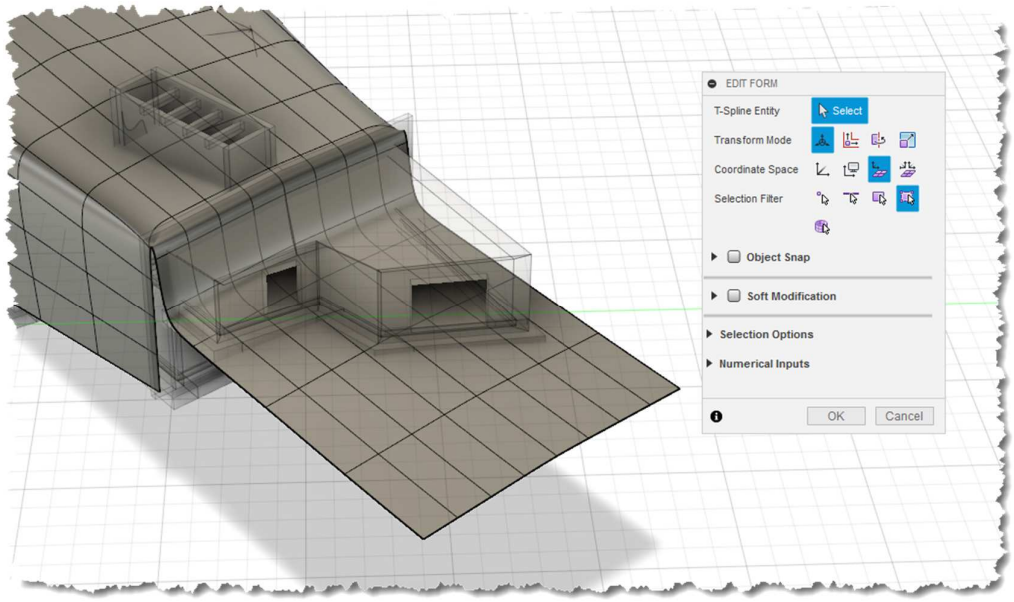
13. The various surfaces are now joined into a single element, check the next picture for reference;



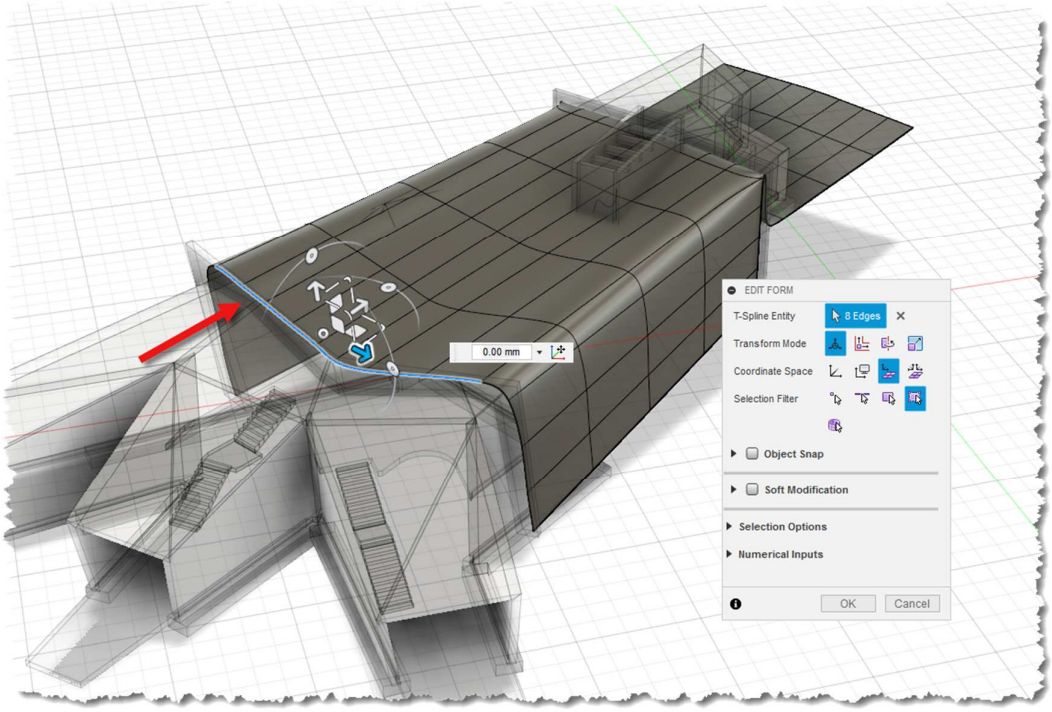
14. Select the indicated edges and click on the “Edit Form” command. Drag the edges while keeping pressed the Alt key to add new faces to the main surface;



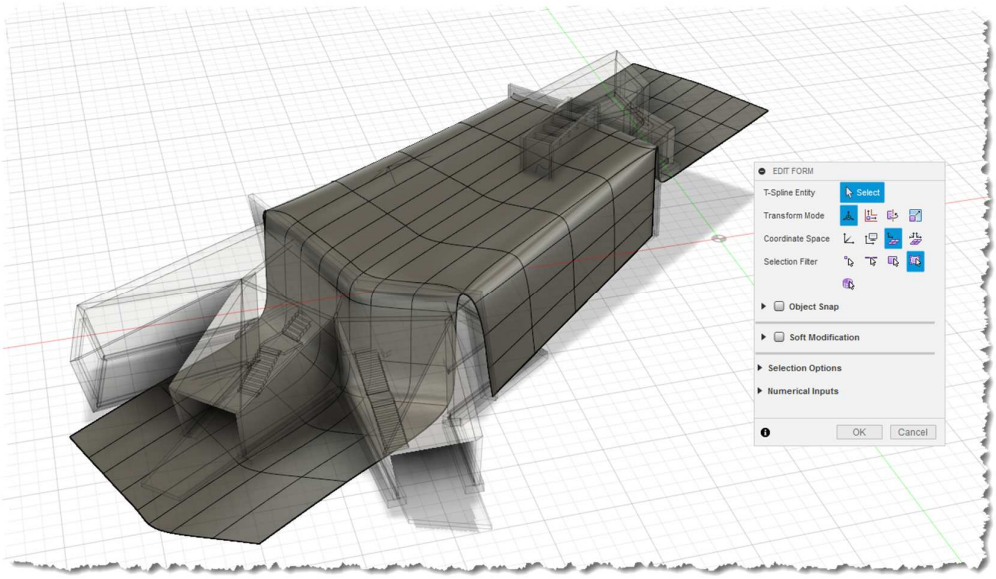
15. Create a “descending” surface in order to match the foundations level on the front part of the building. Take the next image as a reference;



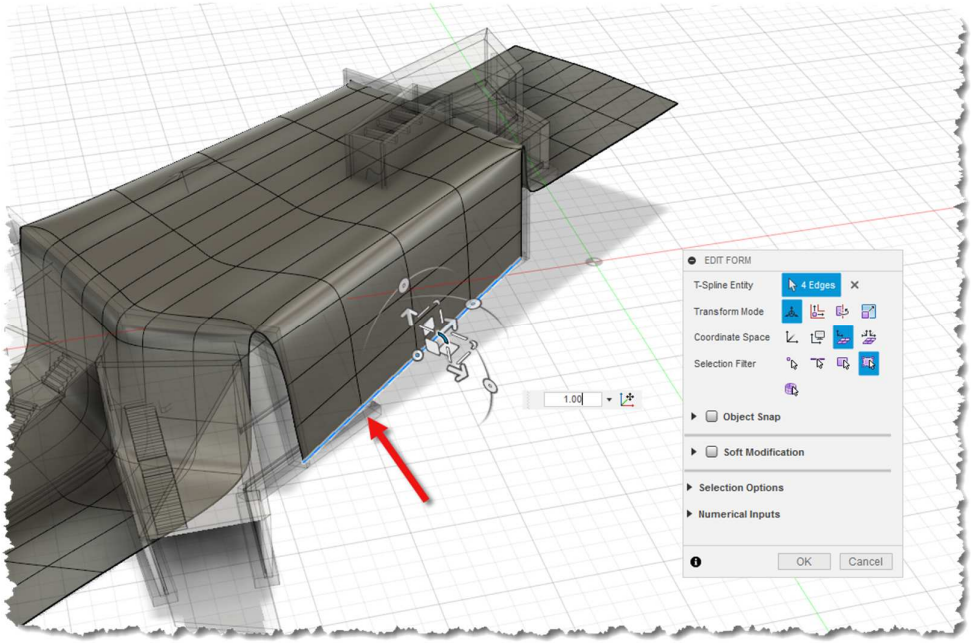
16. Select the top lines of the lower part of the surface and proceed in the same way,



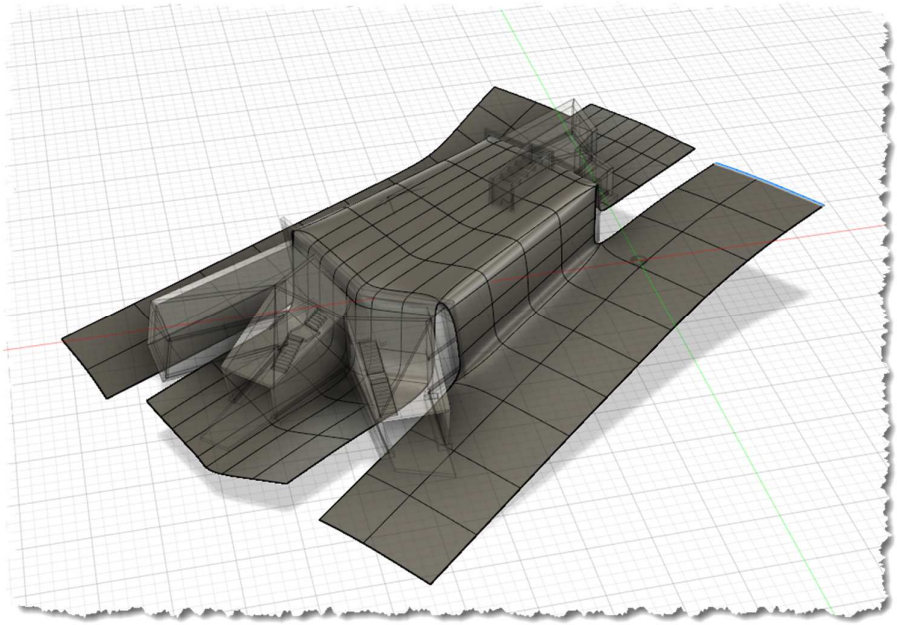
17. Use the “Selection Space” for the “Coordinate Space” setting to simplify the additional surface creation;



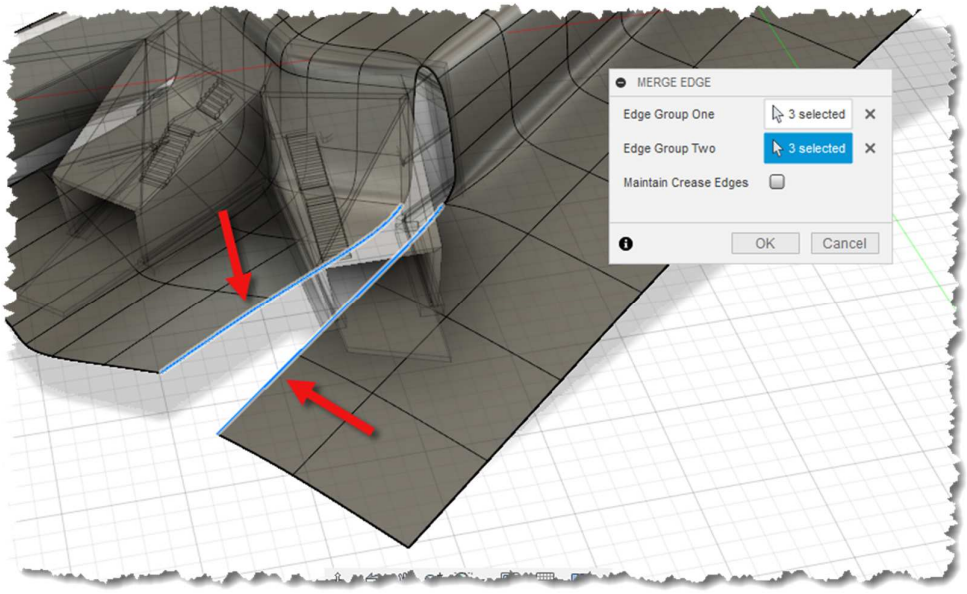
18. Repeat the same procedure for the side edges to expand the toposurface;



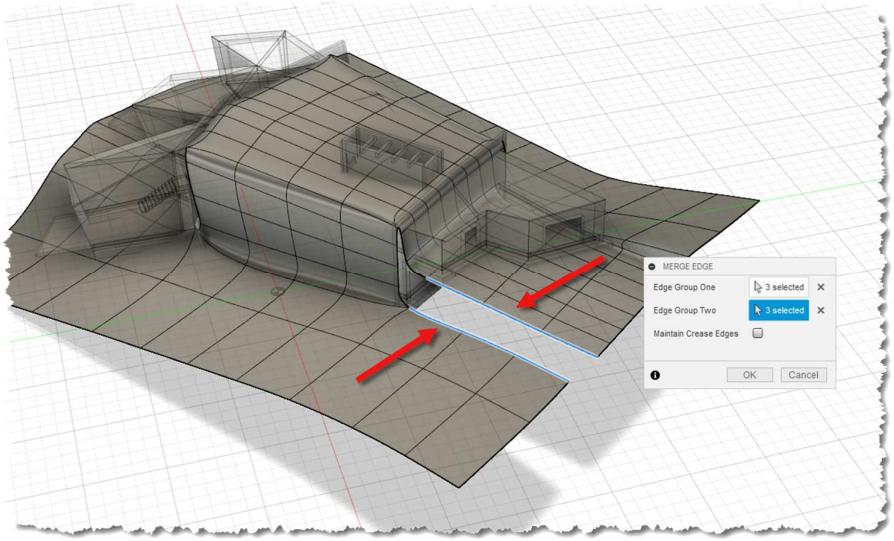
19. Expand the surface edges to reproduce the following picture;



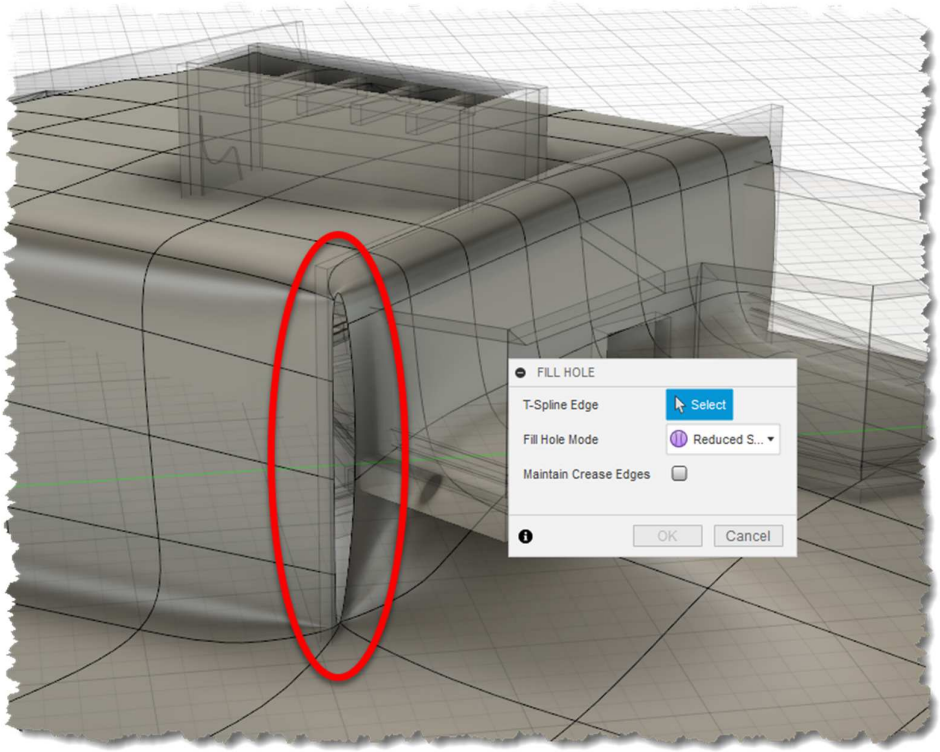
14. Use the “Merge Edge” command to connect the surface strips as shown below;



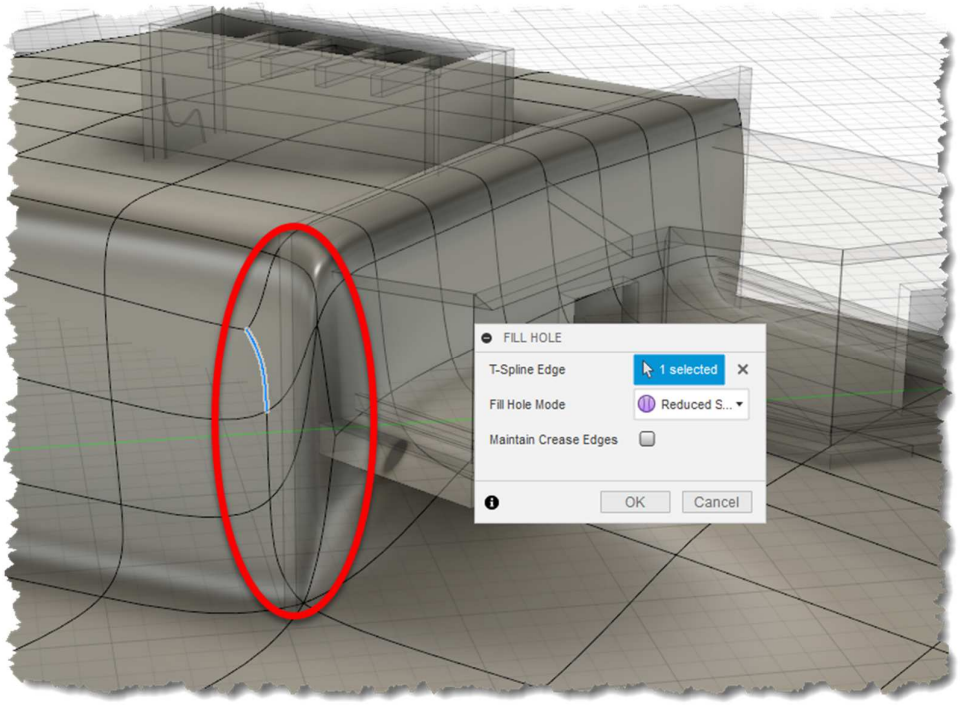
15. Repeat the same for each of the four corners;



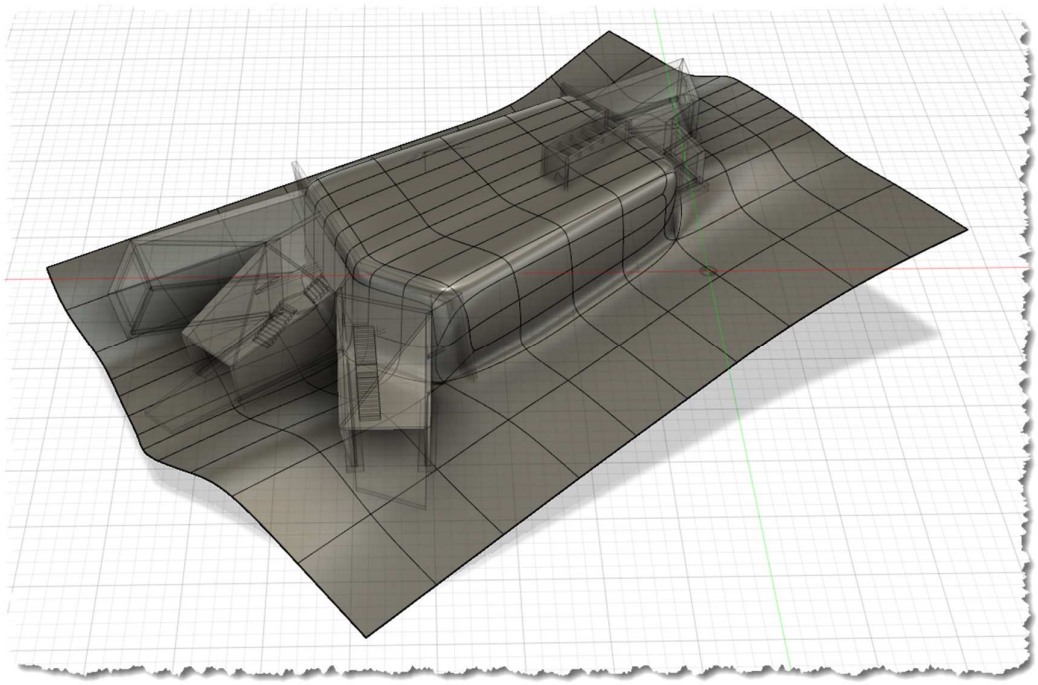
16. The “Merge Edges” command allowed to create a smooth and continuous surface but at the same time four "holes” are remaining on the corners and these need to be fixed. Check below image for reference;



17. Use the “Fill Hole” command under the “Modify” menu of the “Freeform” environment to create a patch connecting the edges of the four openings;



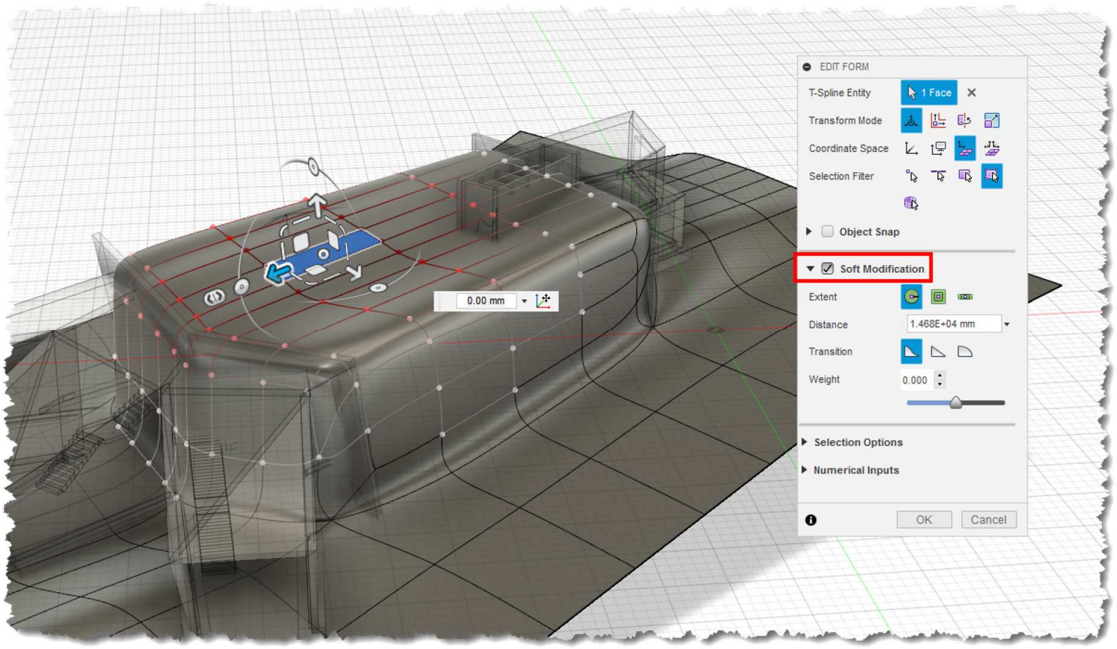
18. The result of this set of activities should have generated a result like that one show below;



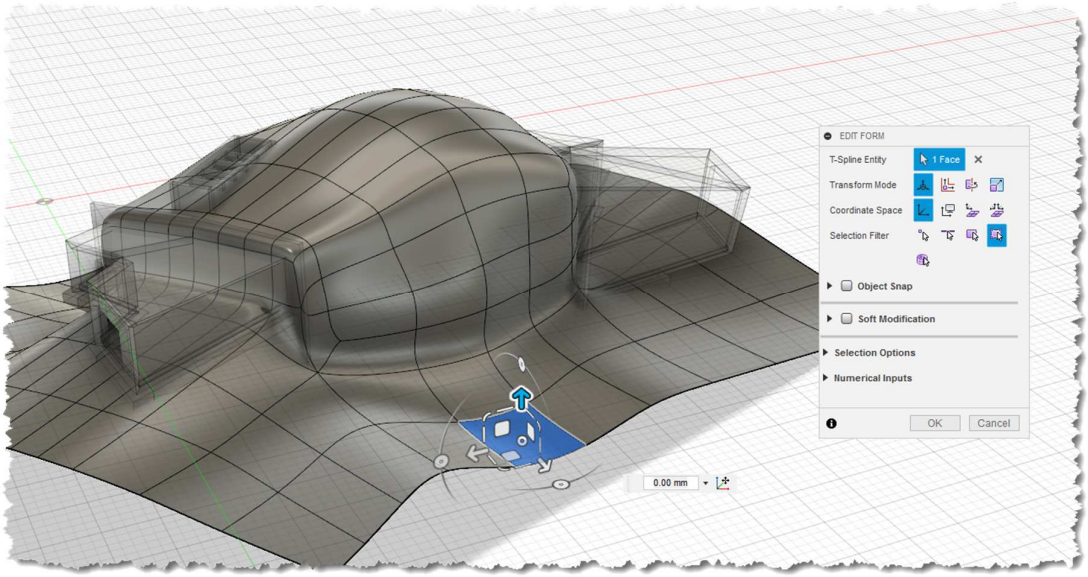
13.2 Surface editing and refinement

The created surface is going to be manipulated to simulate the “hill” above the building and the “cliffs” below the three branches.

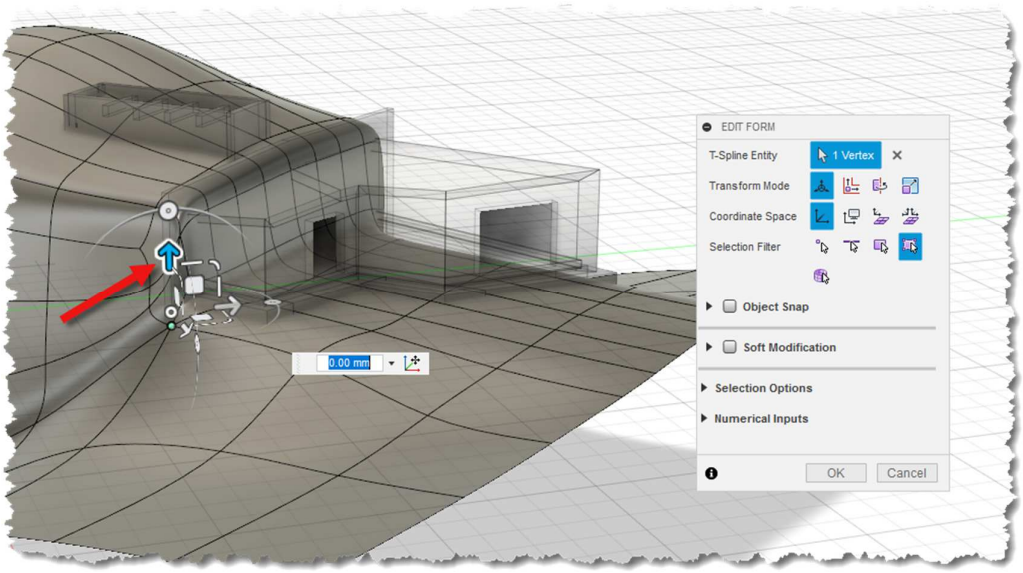
1. Use the “Edit Form” command to move up the desired faces to reproduce the shape of a hill or hump. In this case the “Soft Modification” setting could be an option to obtain a smooth and continuous surface. Test different “Weight” configurations to propagate the edit distance to nearby elements;

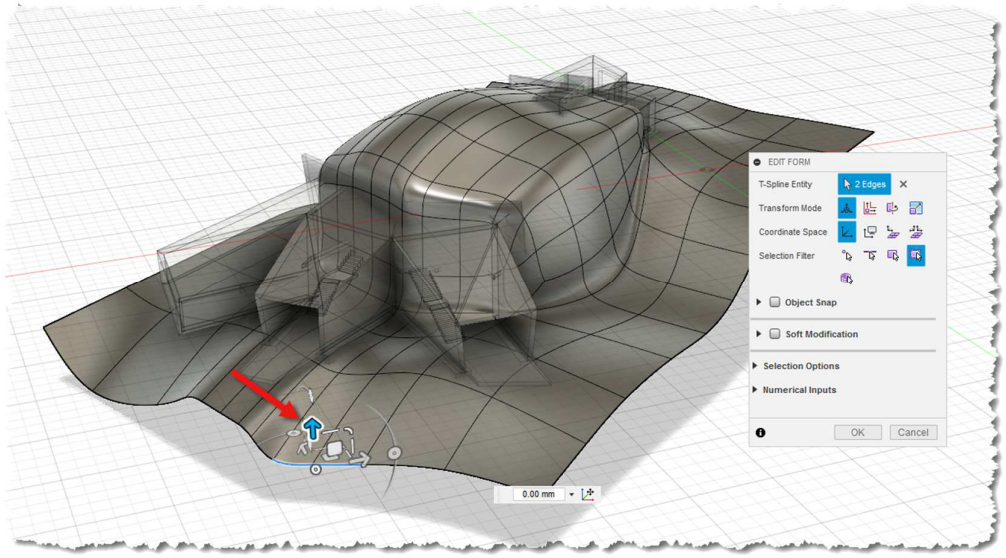


2. Also, modify the side portions of the surface and test different types of “Coordinate Space” settings to see how they affect the model editing;

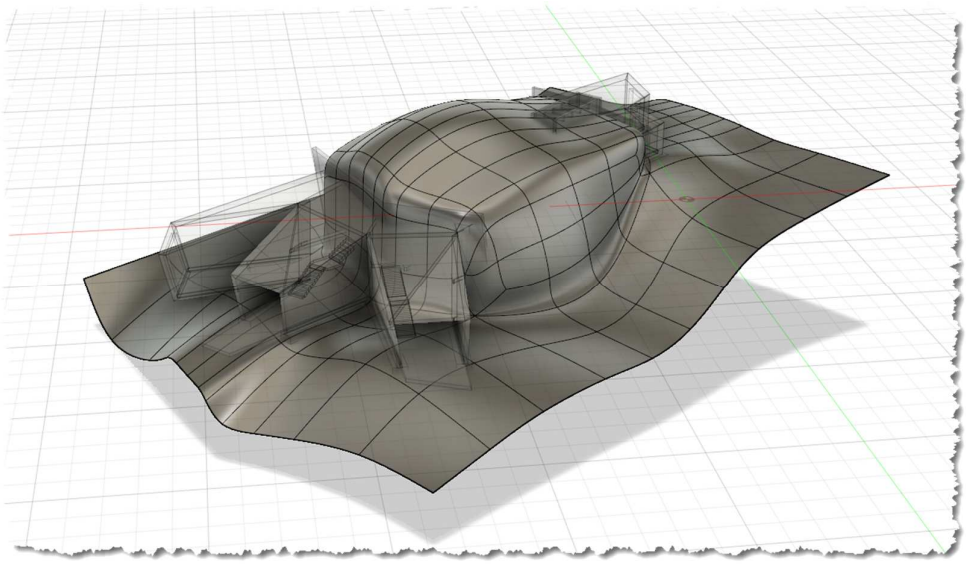


3. With the scope of bringing the surface position at foundations level, vertices or edges may be edited to obtain the desired shape;

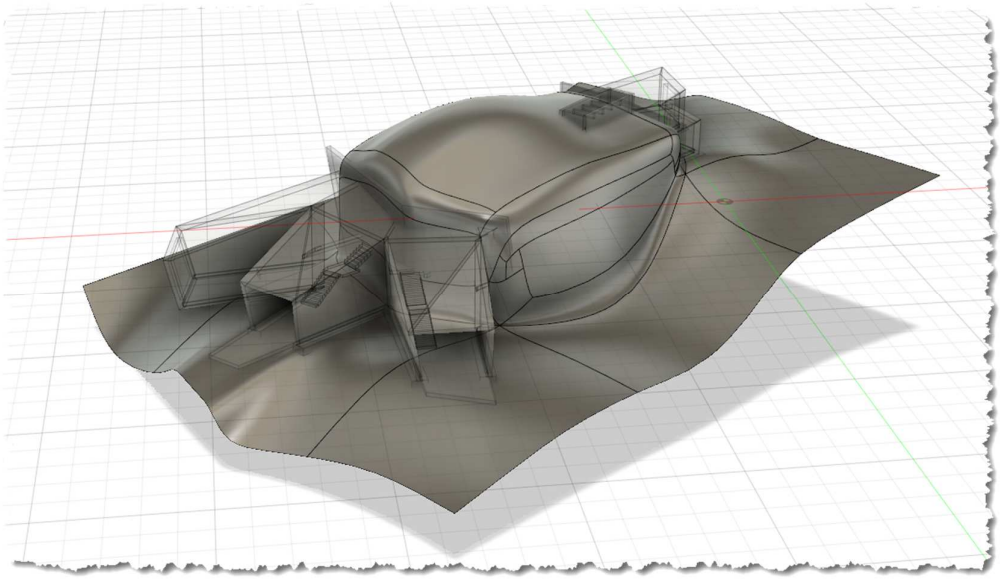




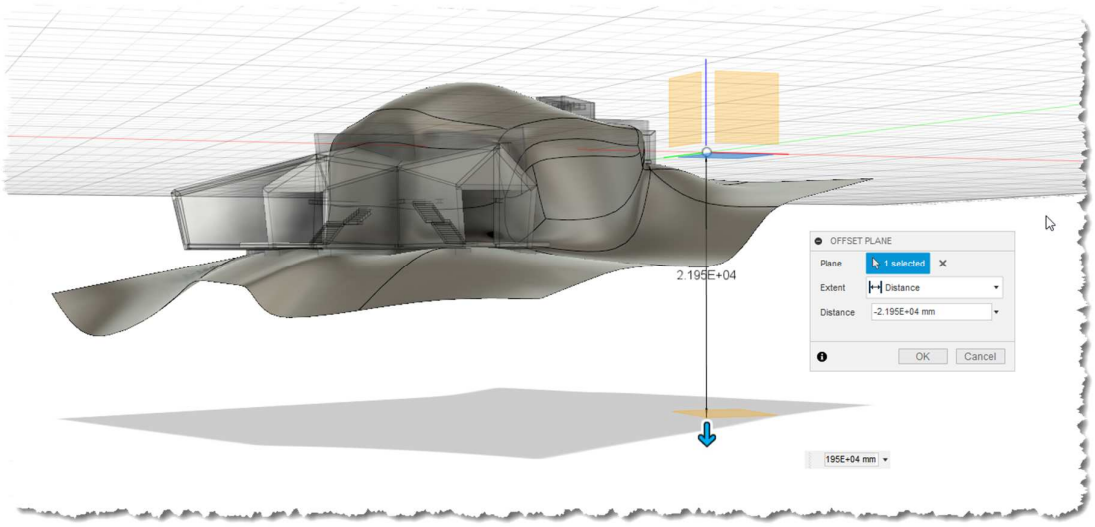
4. Once the surface is configured with the desired shape, click “Ok”.



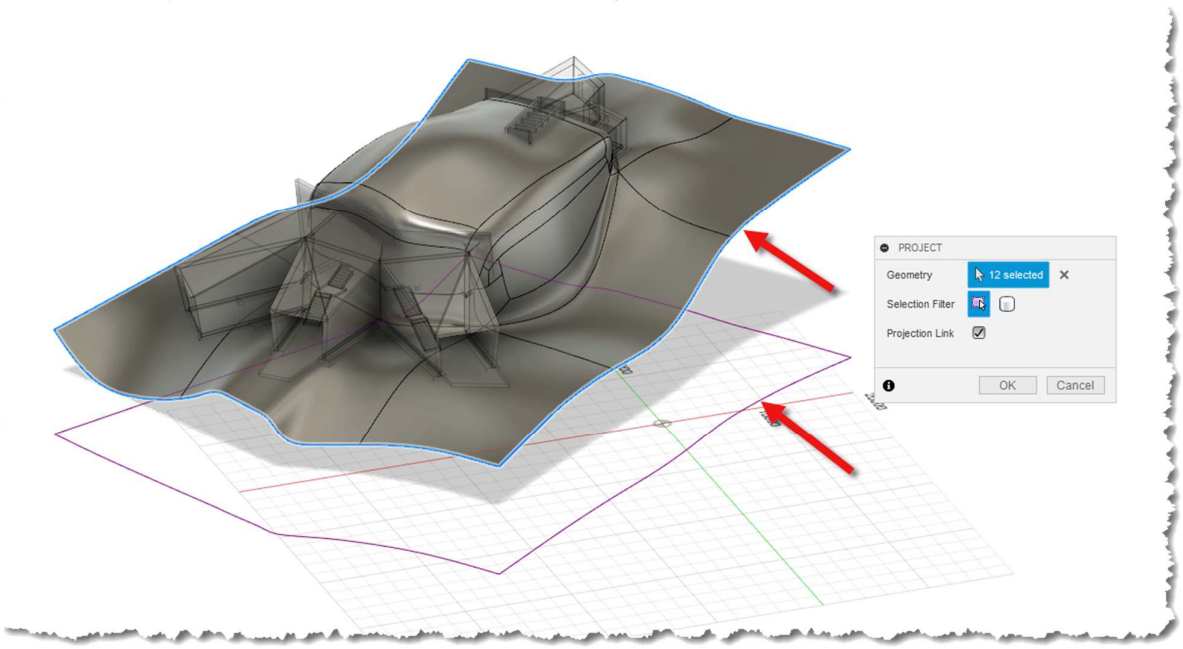
5. Click on “Finish Form” to transform the sculpt object to a standard surface;



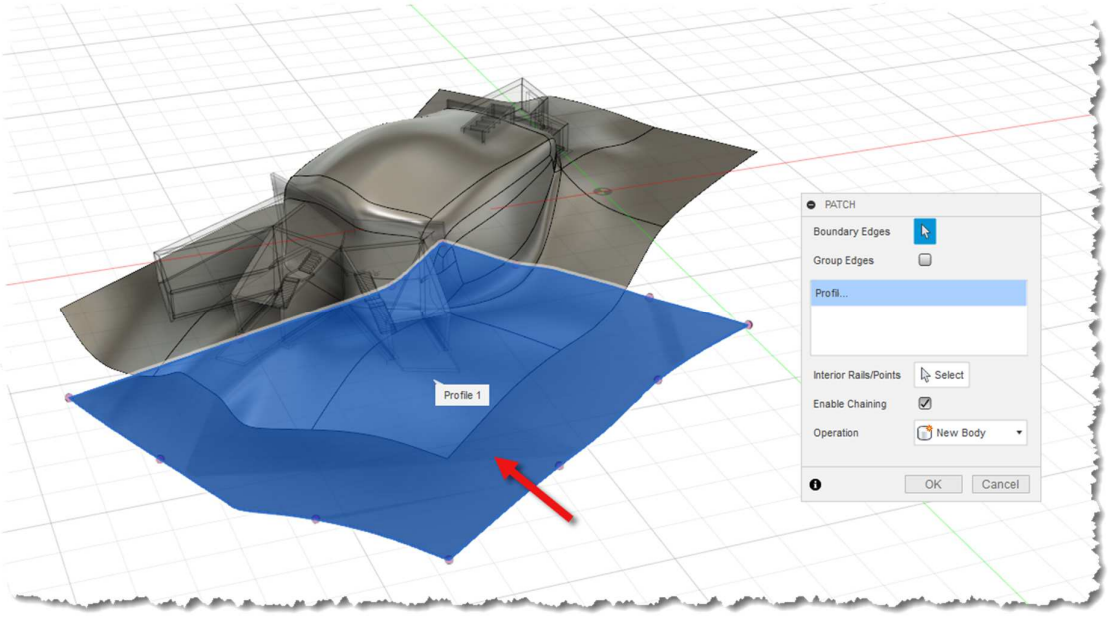
6. A new horizontal surface is going to be created. This will be then connected with the shape representing the topography to obtain a solid body;
7. Create a new reference plane using the “Offset Plan” command; bring it below the existing model;



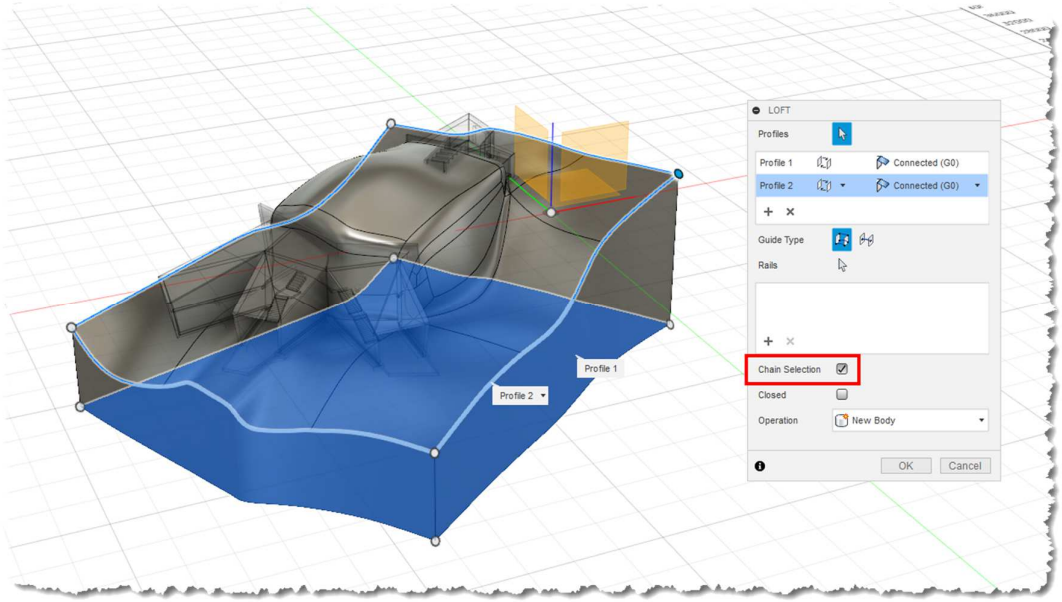
8. Start a new sketch and indicate the new place as the host; use the “Project” command to capture the topography edges projections. Click “Ok” when done;



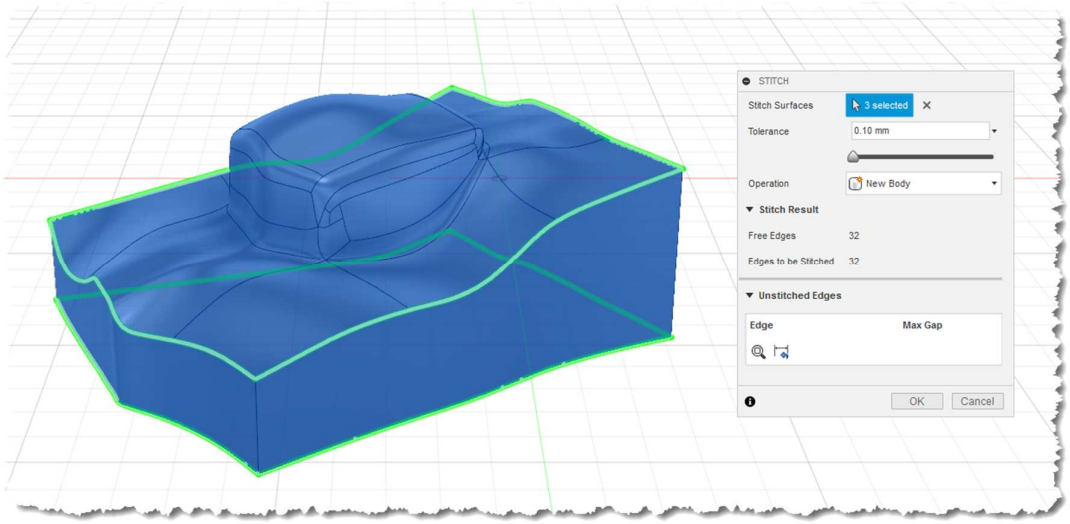
9. Use the “Patch” command inside the “Patch” workspace (this command and the related workspace have the same name in this case). Click “Ok” to create the planar surface;



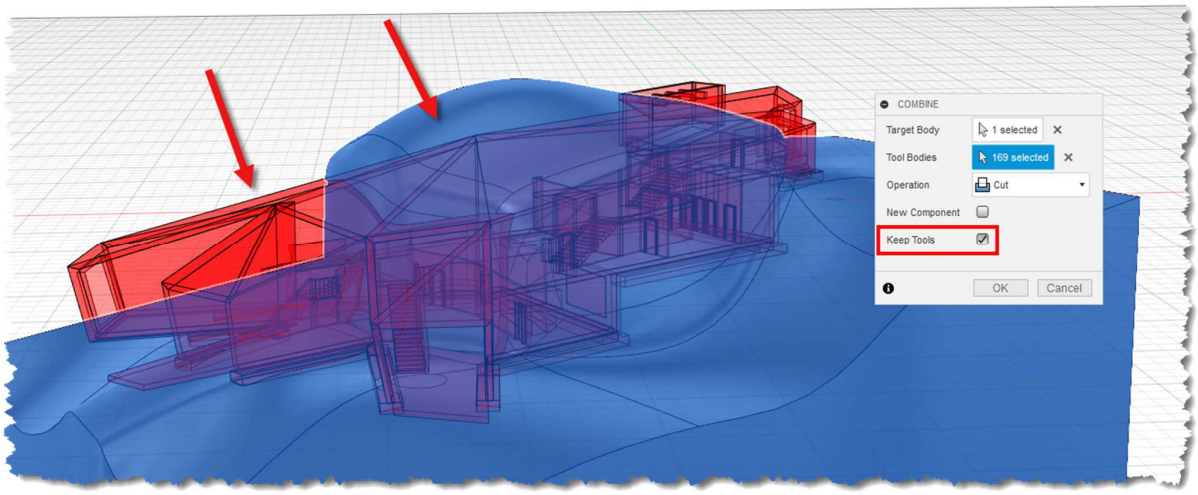
10. Use the “Loft” command inside the “Patch” workspace to connect the lower surface edges to those of the topography. To simplify the edge selection, check the “Chain Selection” option. Click “Ok” to create the connection surfaces between the two objects;



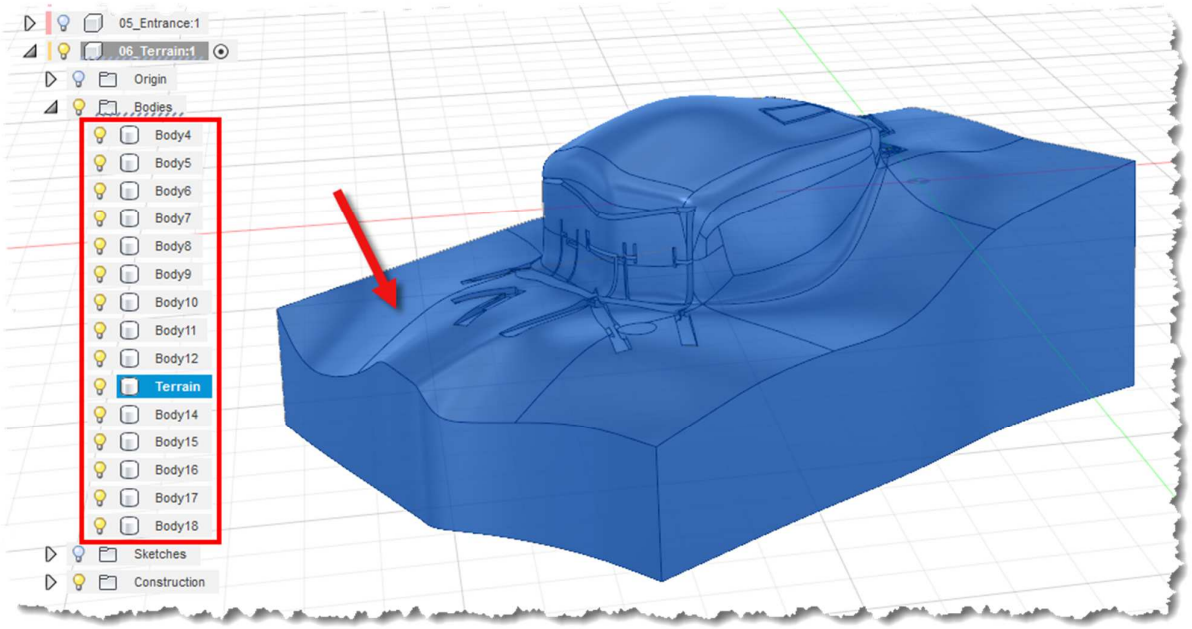
11. Click on the “Stitch” command to join the created surfaces. If those bound a closed volume, the “Stitch” command generates a solid instead of a polysurface;



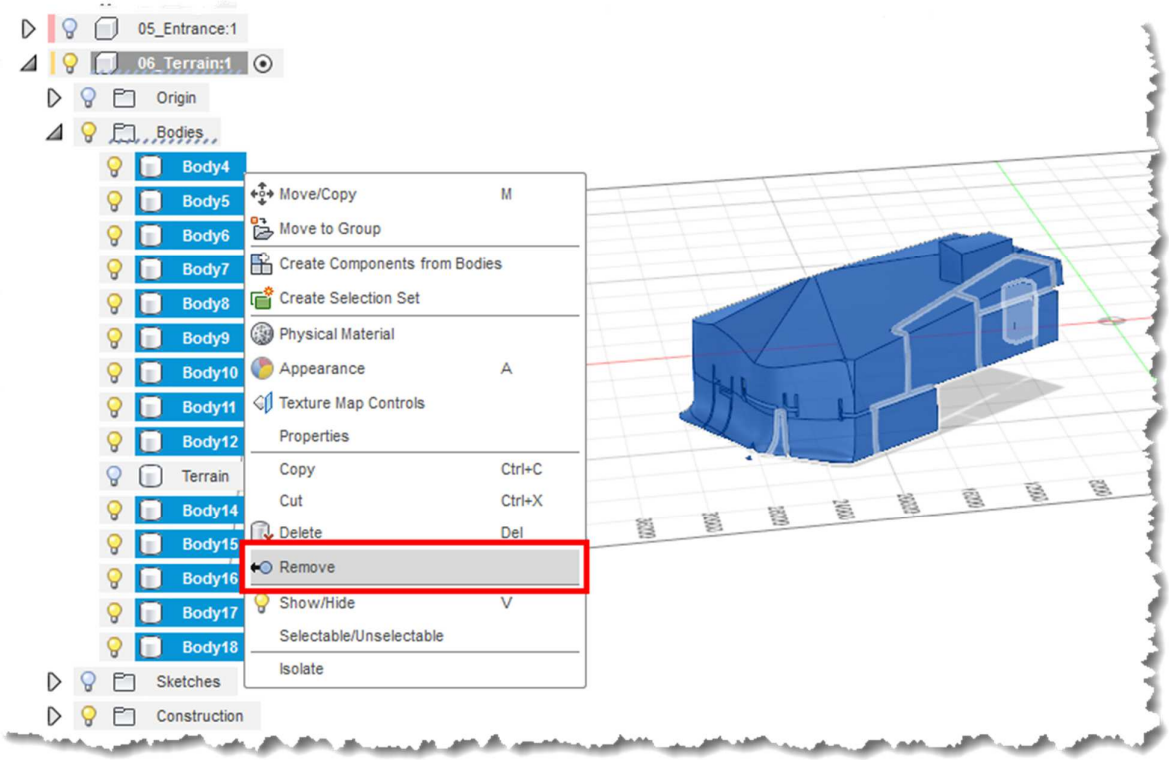
12. The next step is cutting the new volume with the structural component in the model. Switch the “01_MMM_Structure” component on;
13. Use the “Combine” command to cut the topography. Check the “Keep Tools” option to maintain the structural component after the command is executed;



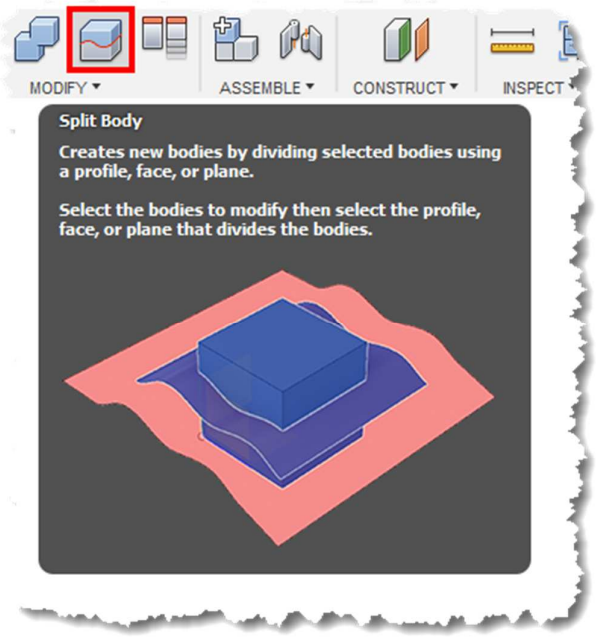
14. Several bodies have been created since the “Combine” command split the toposurface volume in different parts;



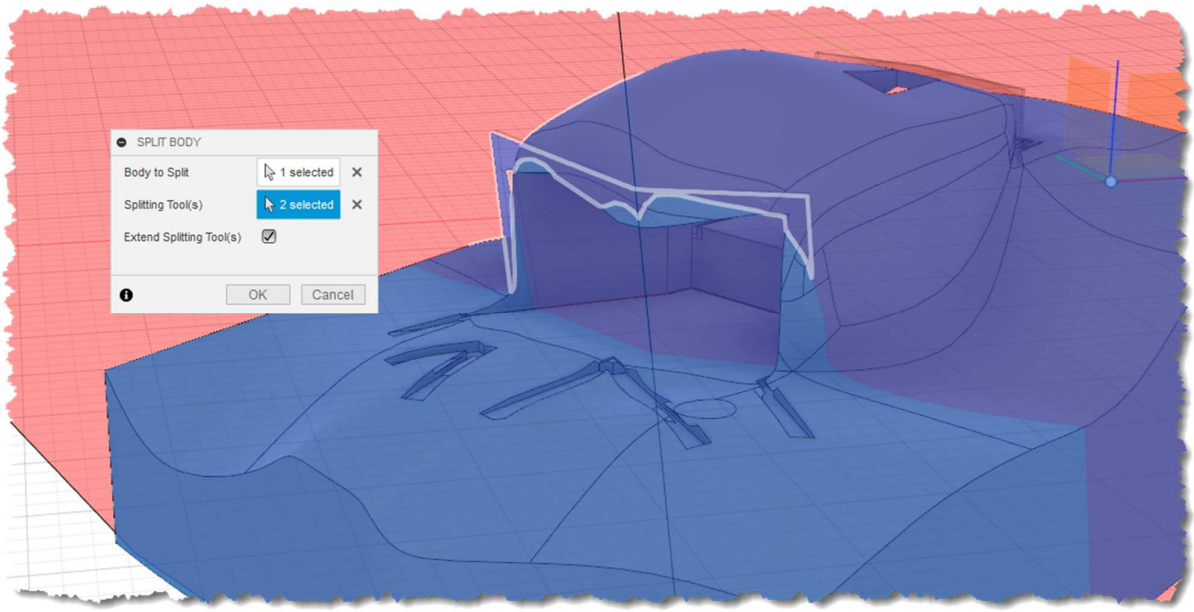
15. Select the body to be maintained and turn off its visibility;
16. In the project tree, select the remaining bodies, right click and choose the “Remove” option;



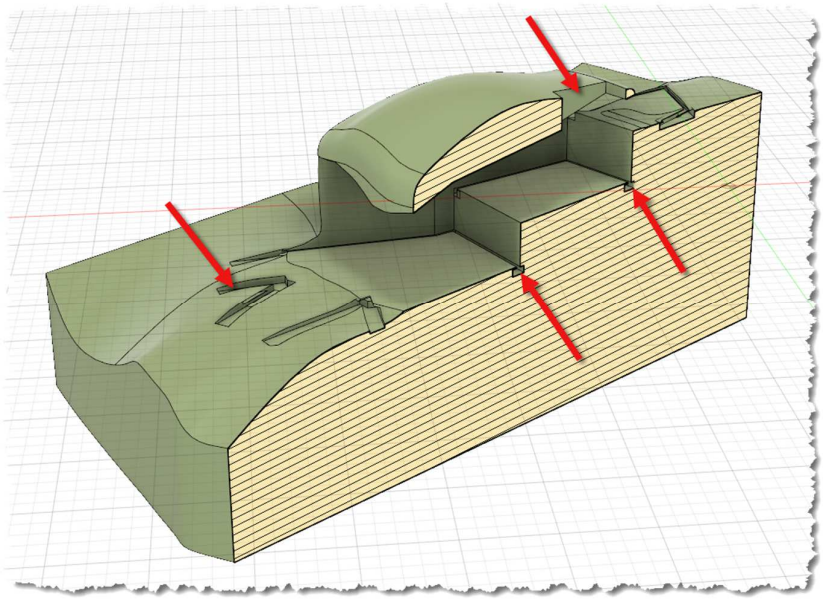
17. This will remove the bodies without affecting the shape or the properties of other elements;
18. If needed, use the “Split Body” command to remove the parts of the toposurface which extend beside the retaining walls

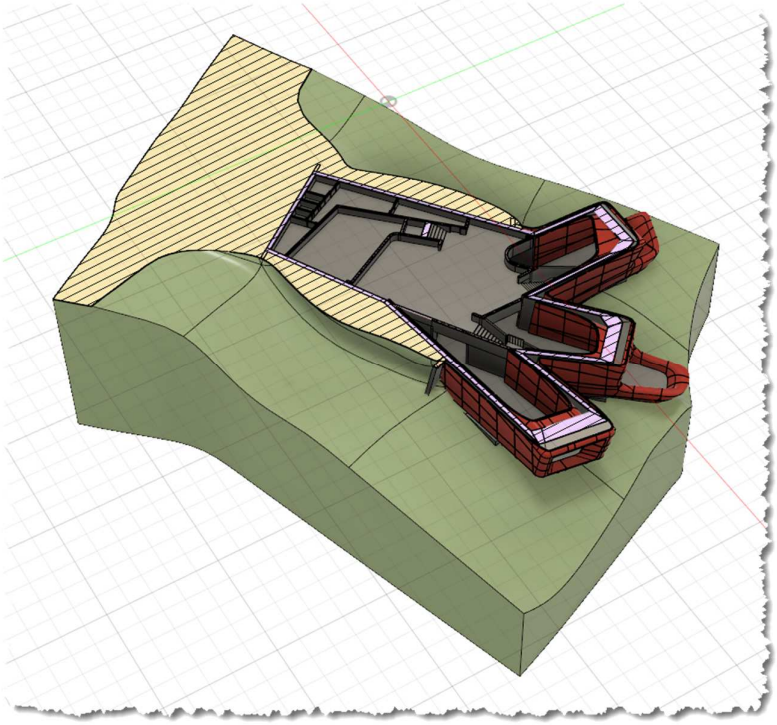


19. For the current example, the main topography is split with the two faces of the lower retaining wall;



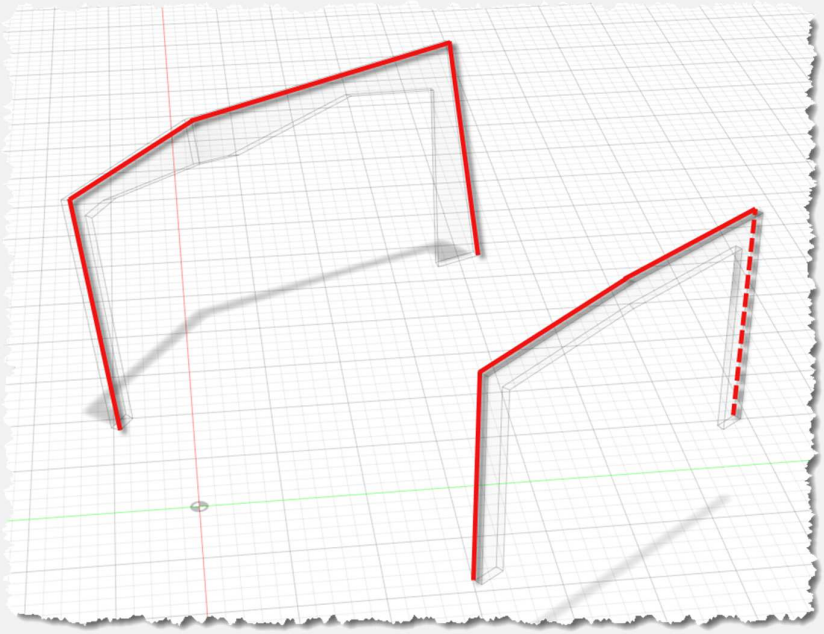
20. The result body is representing a solid topography which brings the footprints of the building components such as foundations, walls and slabs. The generated solid has the same coordinates of the original Revit files so that it can be exported and placed in the aggregated Revit model;



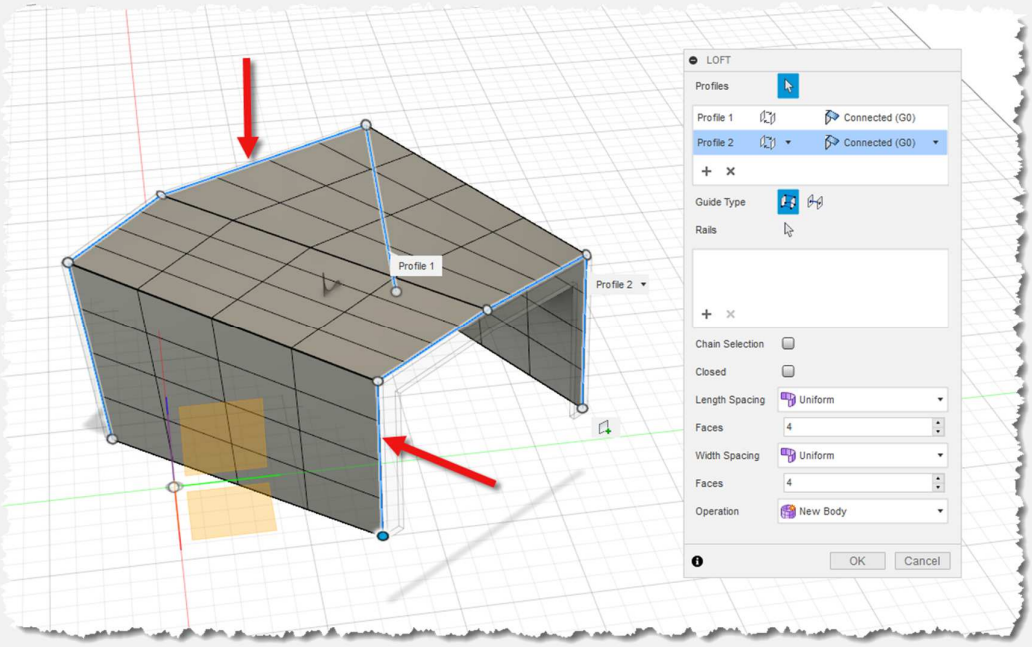


13.3 Alternative method for the main toposurface

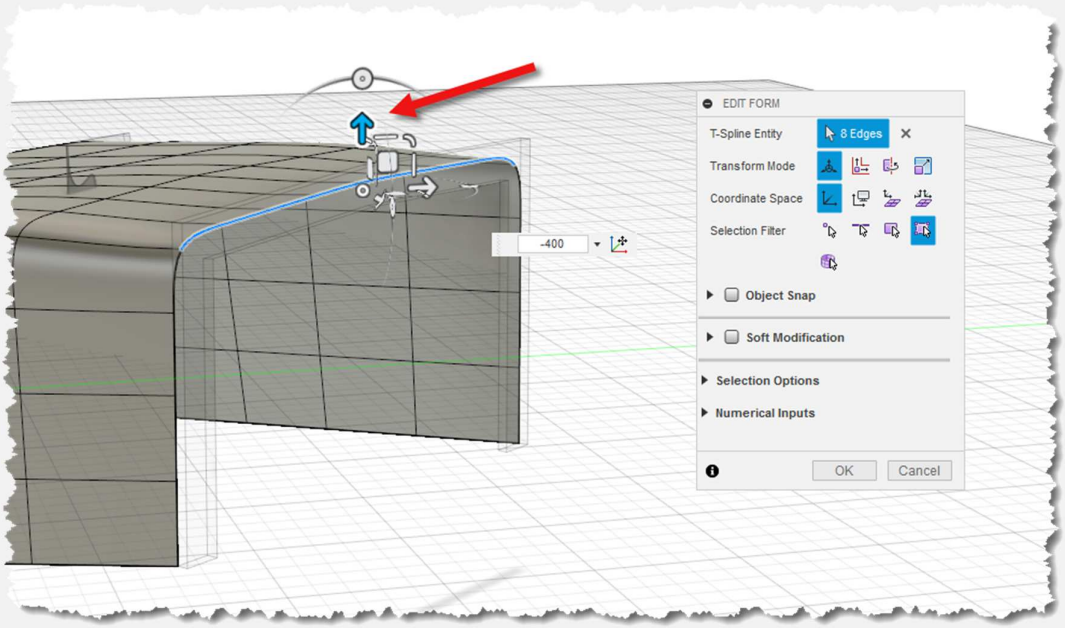
1. Open the “Freeform” environment and select the “Loft” command;
2. Use the identified lines to define the first and the second profile;



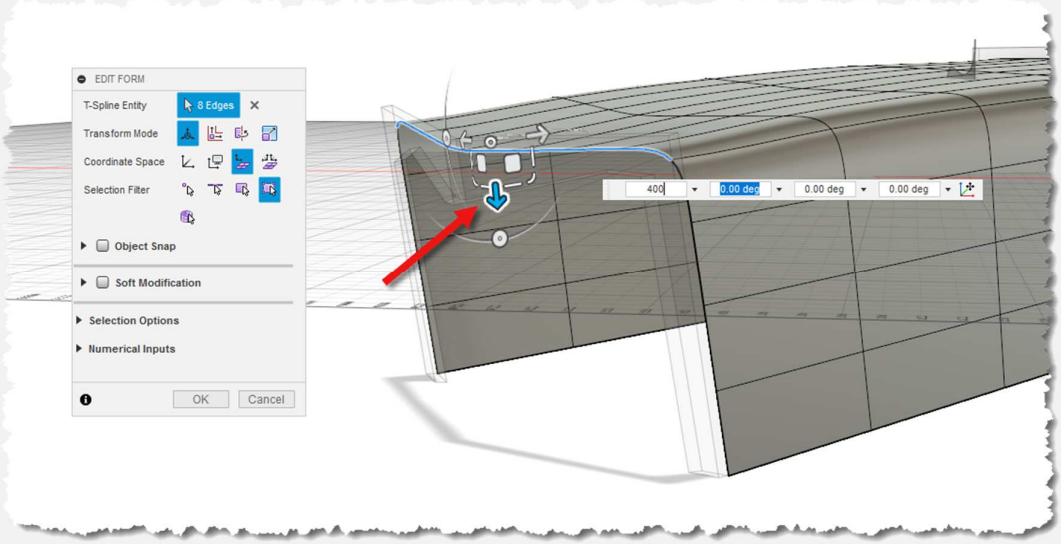
3. Define 4 faces both for the Length and the Width;



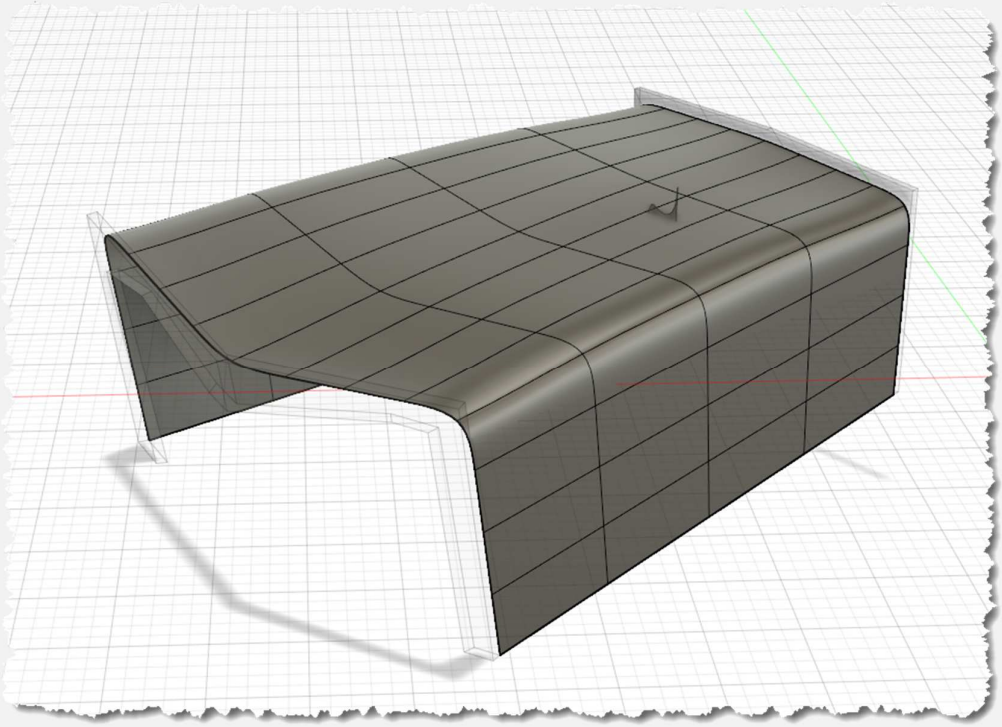
4. Use the “Edit Form” to move downwards the top lines of the first edge; move the edges of 400 mm;



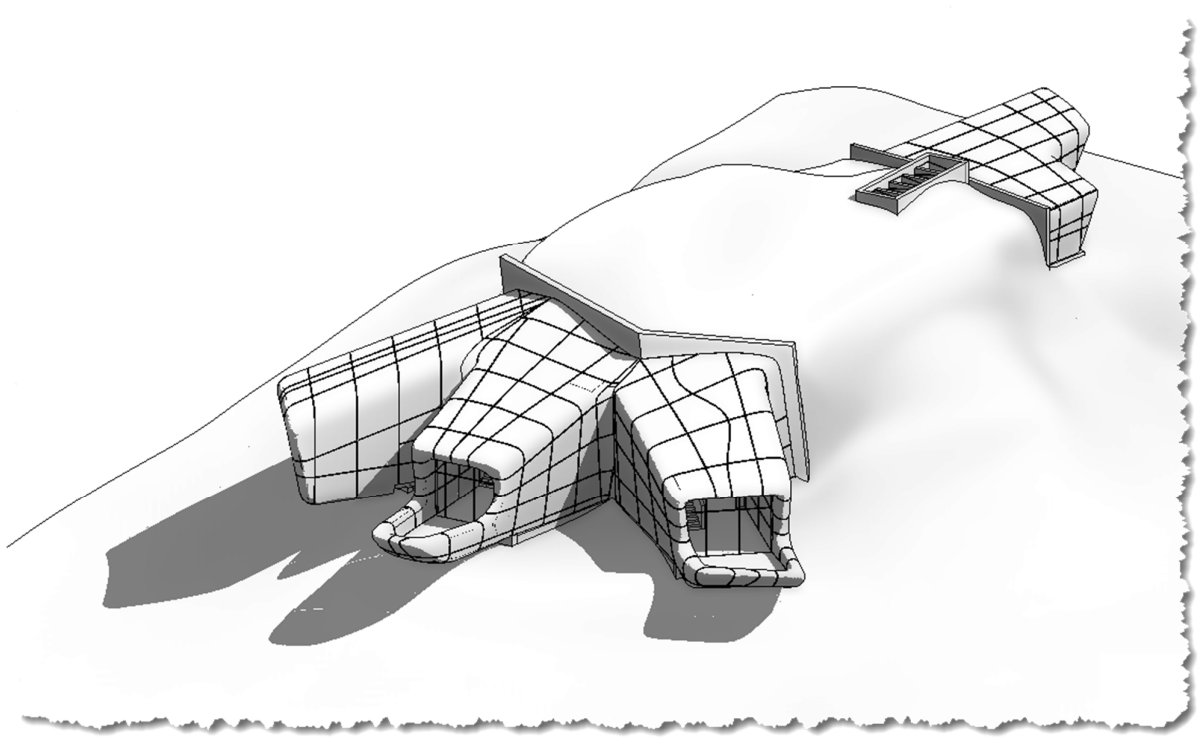
5. Select the top lines of the bottom profile and move them downwards of 400 mm using the “Selection Space” setting for the “Coordinate Space”. This will make the edges sliding along the “host” face;



6. The surface is now ready to be used;



14 Exercise #6: Fusion 360 to Revit workflow



The scope of this exercise is taking the modeled components and use them inside a Revit model. The whole process shown in this document started with the use of a coordinate set of Revit models which have been used as geometric references inside Fusion 360 to create complex geometries and the cladding systems.

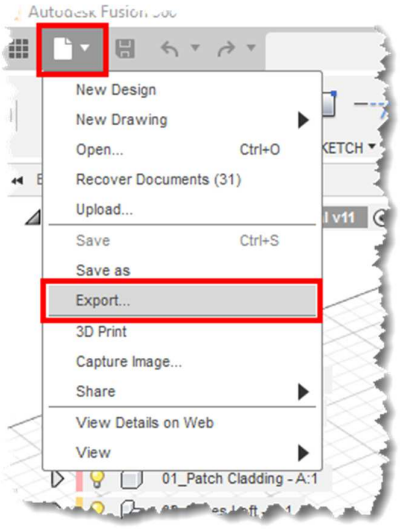
Now the produced material will be exported and used in Revit; two main techniques will be used:

- The use of exported files as they are;
- The use of Dynamo to “translate” exported components to proper Revit families;

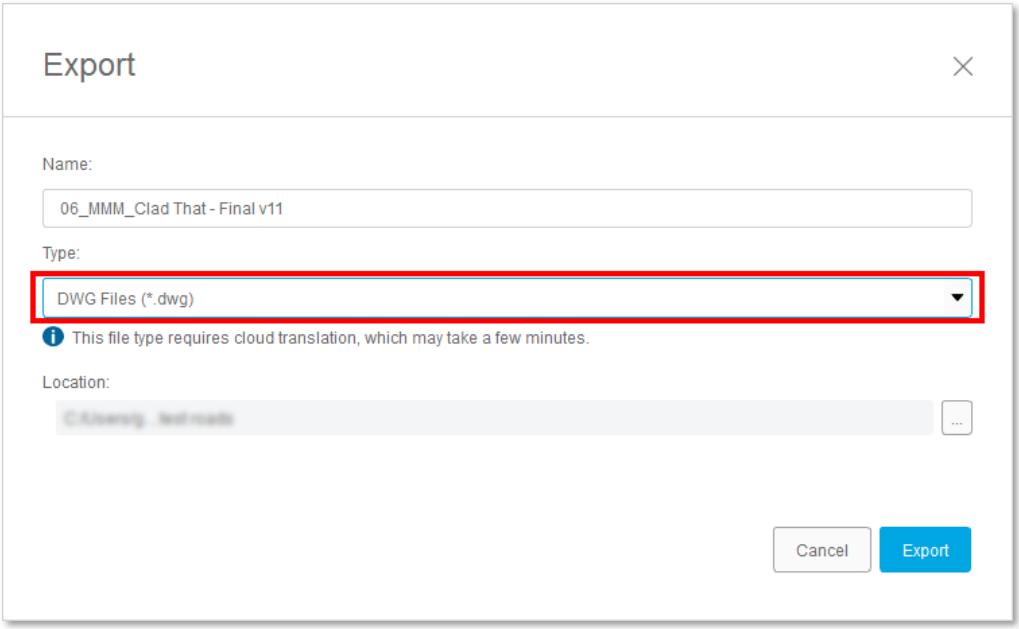
14.1 File Export and Preparation

Fusion 360 allows to export the model or parts of it by using two different approaches.

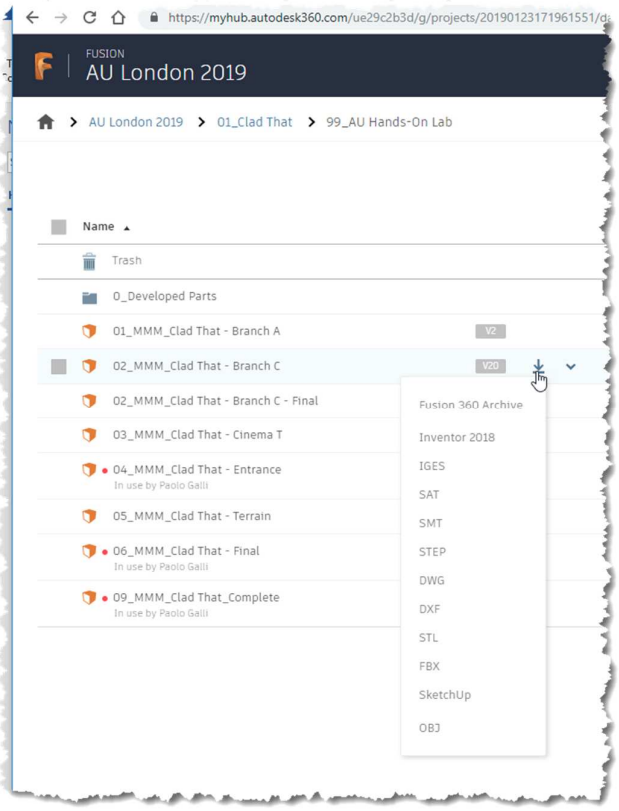
The first one is the export of the entire model using the “Export…” command located in the dropdown menu at the upper left corner of the software interface. It is important to repeat that the whole model is exported this way.



This allows to choose between several file formats and in some cases, this requires a “cloud translation” so that the exported file will be downloaded once it is ready.

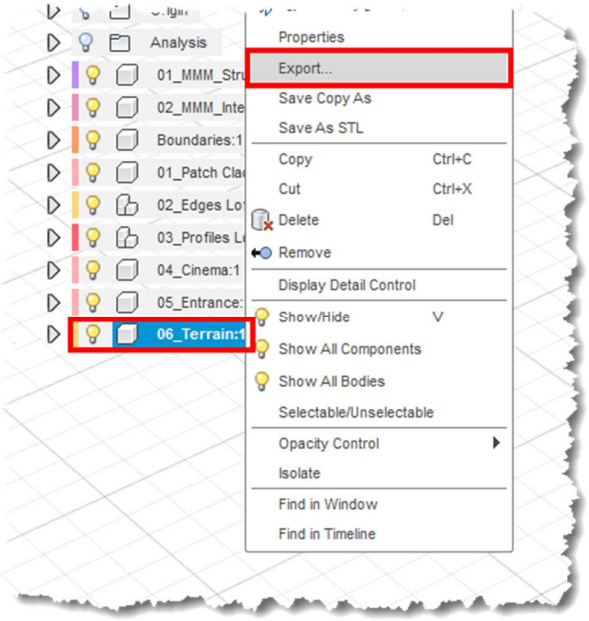


It is also possible to export the whole model using the A360 web interface. Identify the file to be exported and click on the down arrow to choose between the available file format. When the file is ready, a notification is sent by email also providing the download link.

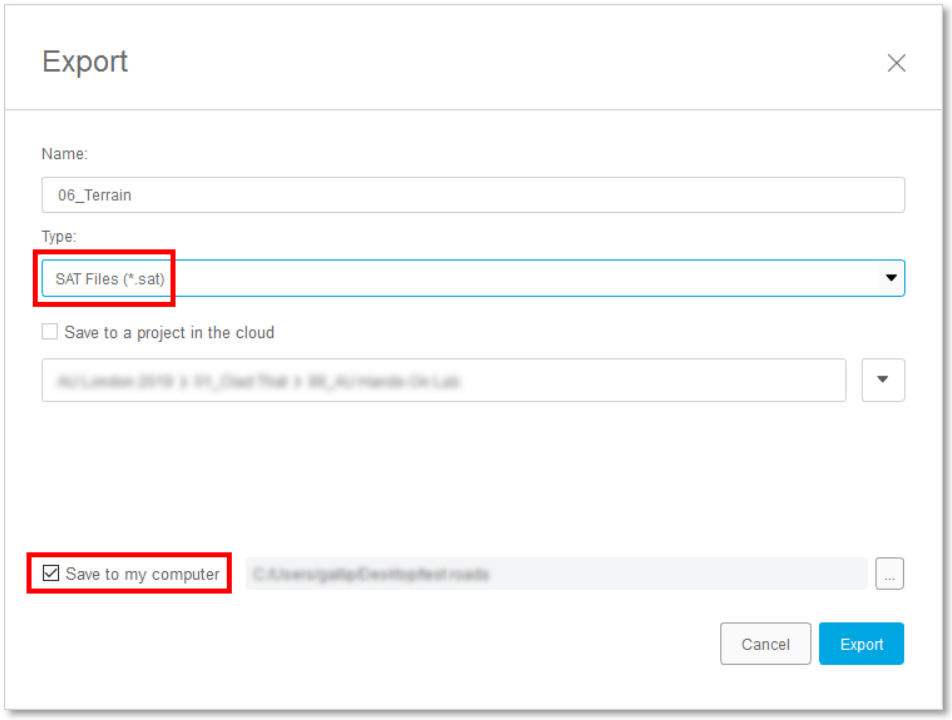


The second method is for exporting single components.

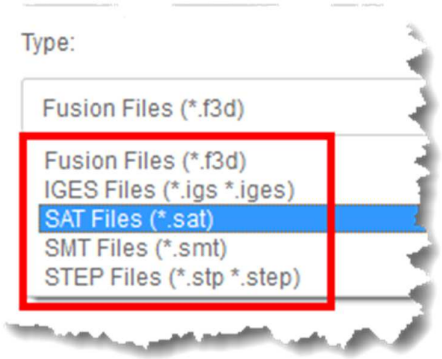
1. Right click on the "06_Terrain" component (for instance) and choose the "Export…" command;



2. Specify “SAT” as the file type and check the option “Save to my computer” then click “Export”;



3. Only a limited amount of file formats are available with this approach;

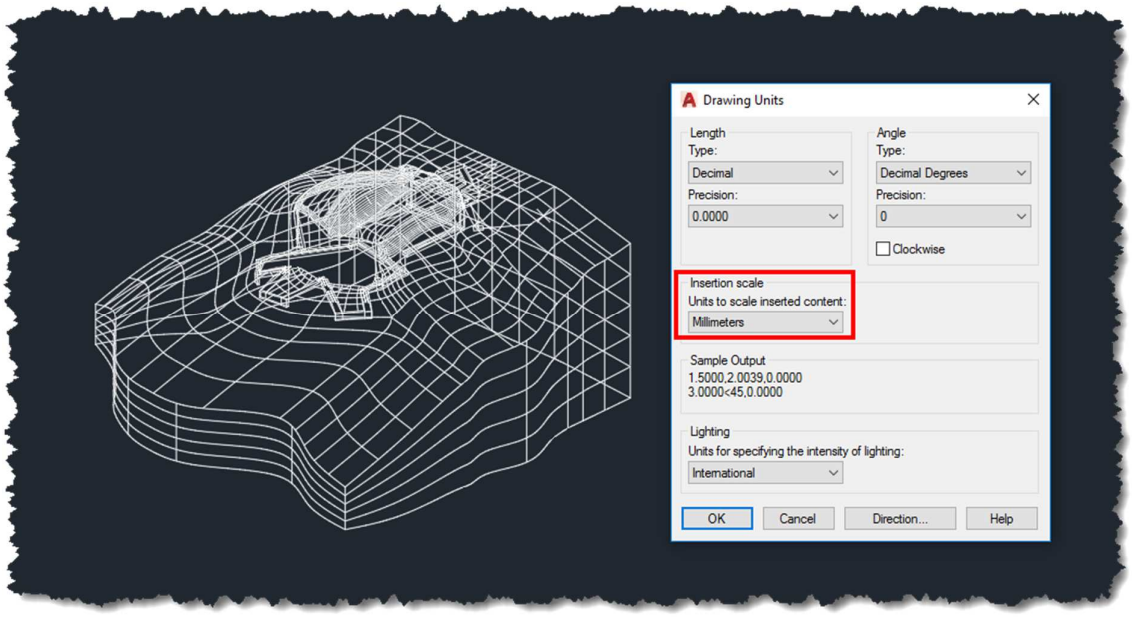


4. Note that this method exports all visible elements inside a component, so be sure to turn off the visibility of un-wanted bodies and elements.

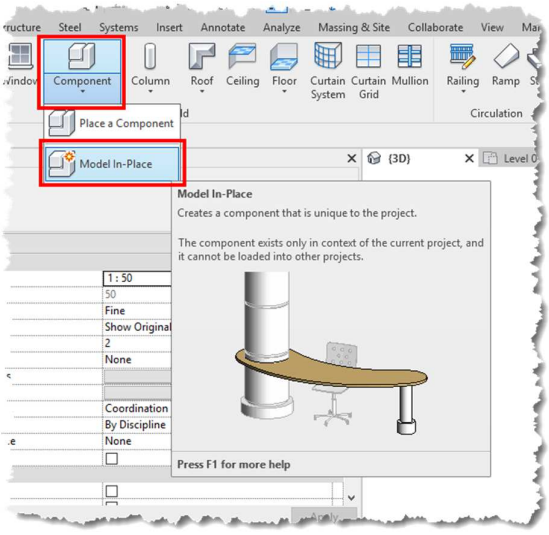
14.2 Use of exported components in Revit

When a component is exported, this can be used into a Revit file. AutoCAD is here used as a "bridge" to prepare the export files to be used inside Revit.

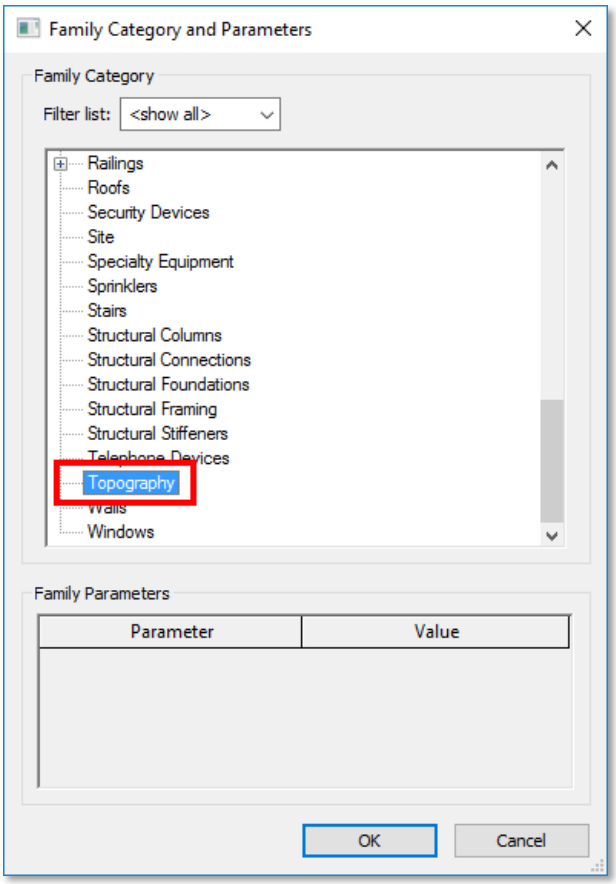
1. Export the "06_Terrain" component as SAT file;
2. Open AutoCAD and import the produced SAT file and export it as DWG. Be Sure to check the consistence between the model size and the file units;



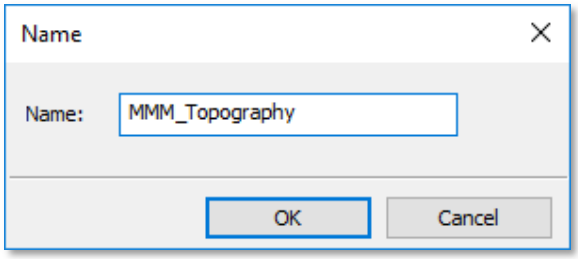
3. Save the file as DWG;
4. Open Revit (it is possible to use the Structural file or another project file in which the structures have been linked origin to origin to remain aligned with the same coordinates);
5. Create a new in-place component;



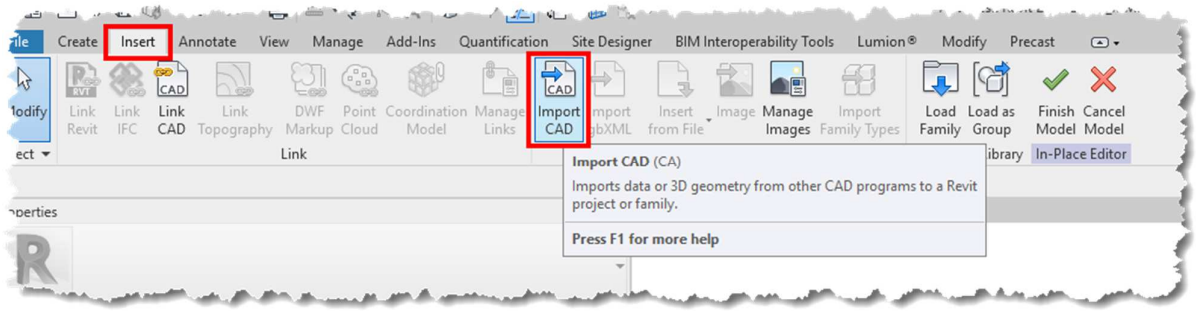
6. Specify the “Topography” category;



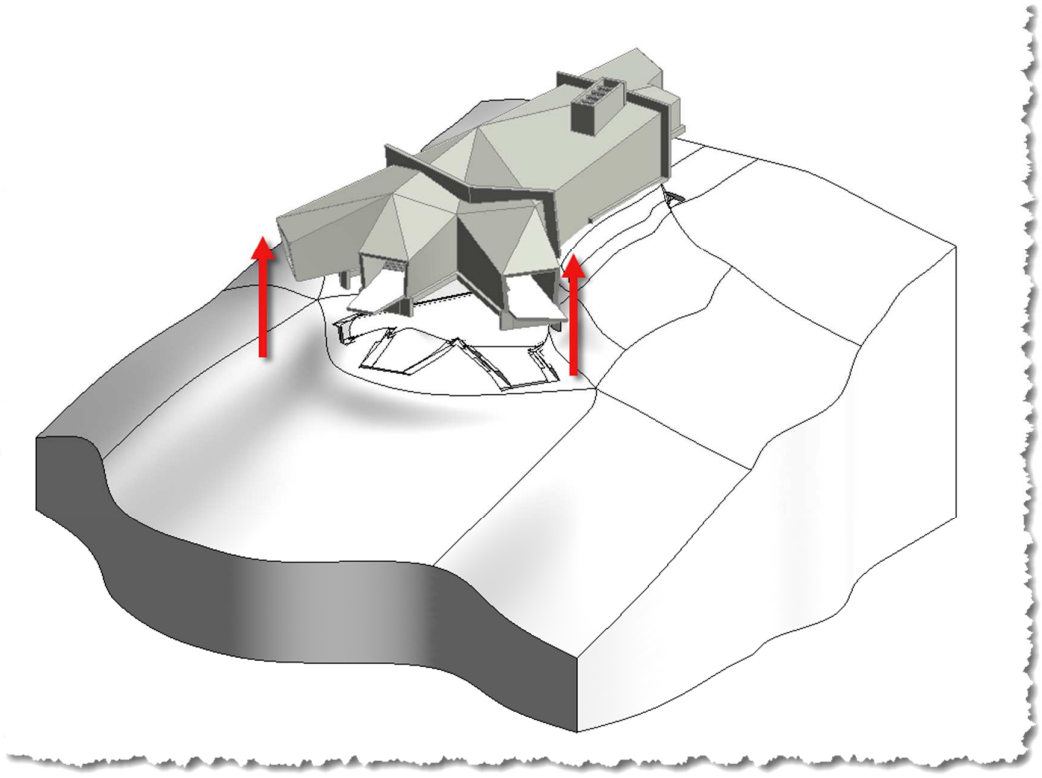
7. Call the in-place family “MMM_Topography” and click “OK”;



8. Select the “Import CAD” command and look for the created DWG file;

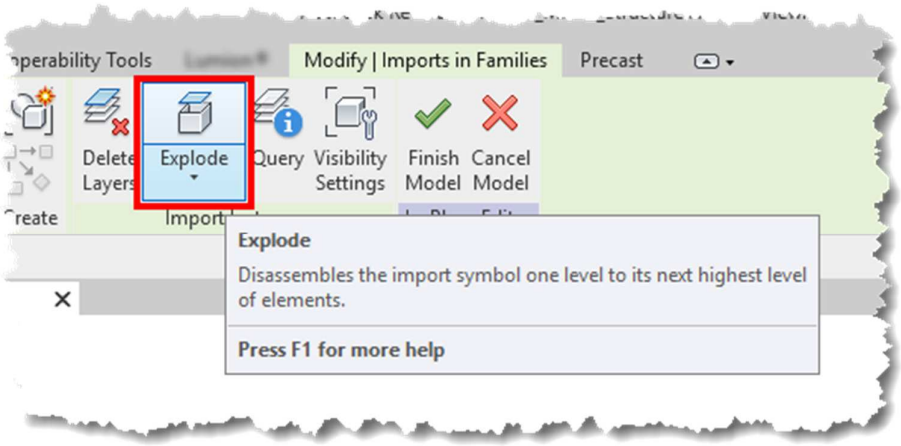


9. Specify “Origin to Origin”;
10. The imported model must be moved to the right elevation since Revit places it at the lower available level if the import command is executed while a 3D view is the current one;

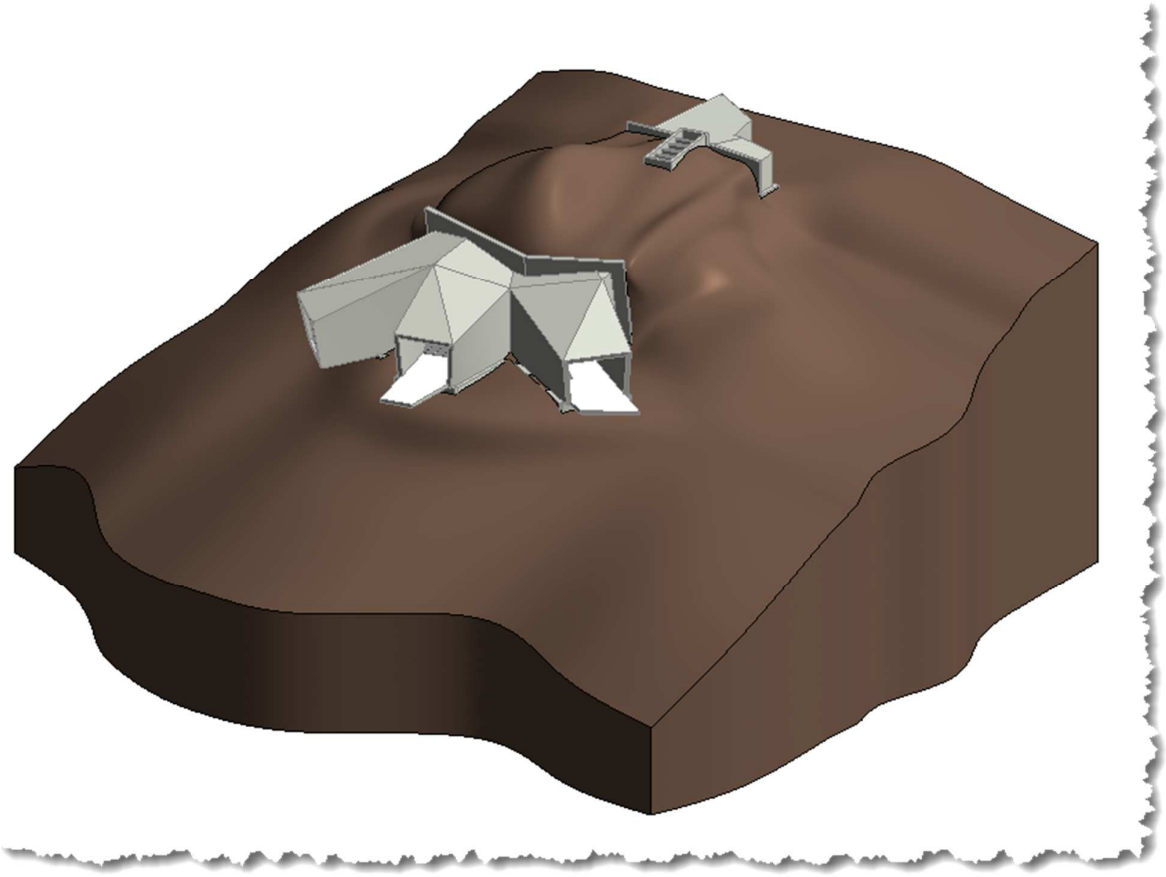


11. Select the topography and specify “Level 0” as the “Base Level”;

12. Select the imported file and then click on “Explode”; this converts the imported solid DWG to a *native* Revit solid. This means that it is possible to apply material parameters to the geometry and modifying it with Boolean operations, for instance;



13. The exploded element automatically takes the “By-Category” material specified for the Topography;



14. Finish the family creation by clicking on “Finish Model”;
15. The created family can now be used in schedules and for drawing production purpose;

NOTE

Transforming a SAT file to DWG using AutoCAD, gives several benefits:

- *It can be edited in AutoCAD, filtering the set of elements to be transferred to Revit;*
- *If imported into a Revit family, the DWG can be exploded and recognized as a native geometry; this means material parameters can be assigned and drawing production is made easier in terms of graphic consistence and quality;*
- *When a DWG is exploded in a Revit family, this can be modified using void or solid geometries performing Boolean operations;*
- *If a DWG is exploded inside a Revit family, this inherits the host family properties. This means that a model imported inside an in-place roof family, will work as a real roof: walls can be attached to its lower face;*
- *Walls from exploded DWG can bound rooms, although in some cases this can work in unexpected ways so be careful!*
- *DWG files can be exploded only if imported; this does not work with linked files.*

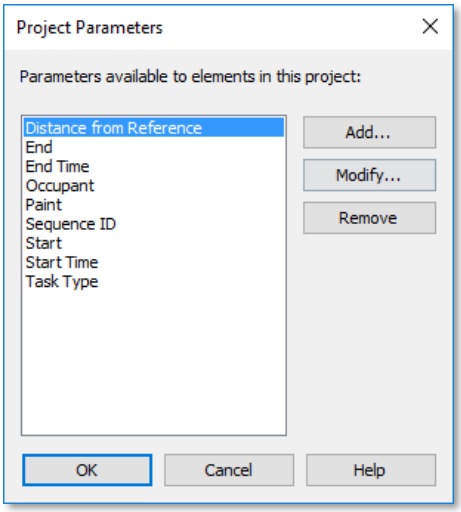
If there is not the need to explode the DWG, SAT files can still be used; the linking is more advisable than importing for a matter of file performance and size. Moreover, the linked SAT files inside an in-place mass family allows to divide its surfaces and place curtain wall panels pattern based while imported SATs do not allow this useful feature.

14.3 Use of Dynamo for the cladding placement

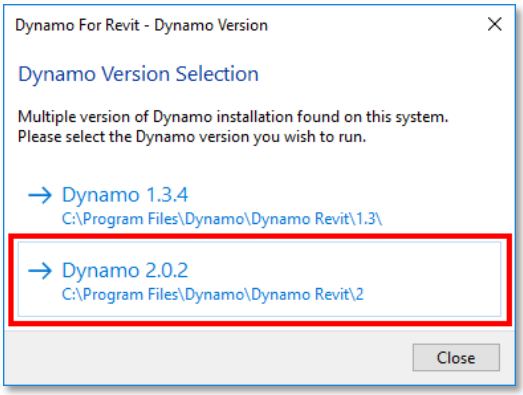
The second method to use Fusion 360 elements in Revit is using a Dynamo script to have all the objects inside the exported file, recognized as single editable Revit families.

Use the provided "03_MMM_Patch Cladding.RVT" file since this has been configured with the following points:

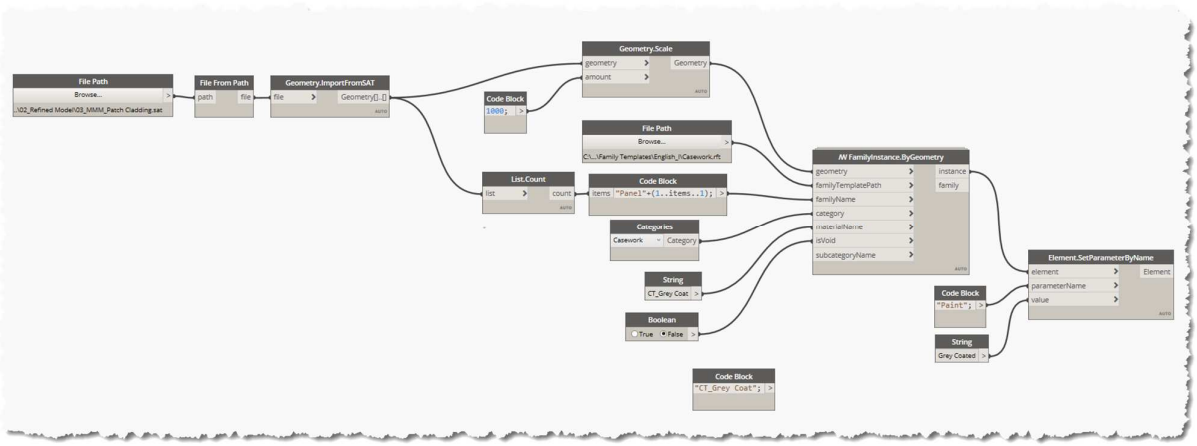
- Coordinates;
- Structural and interior files have been linked;
- Levels have been copied and monitored;
- Several parameters have been setup to be able to schedule the cladding panels and to give an input for the 4D sequencing;



1. In Fusion 360, export the component "01_Patch Cladding – A" in SAT;
2. From the "Manage" ribbon, start Dynamo 2.02;

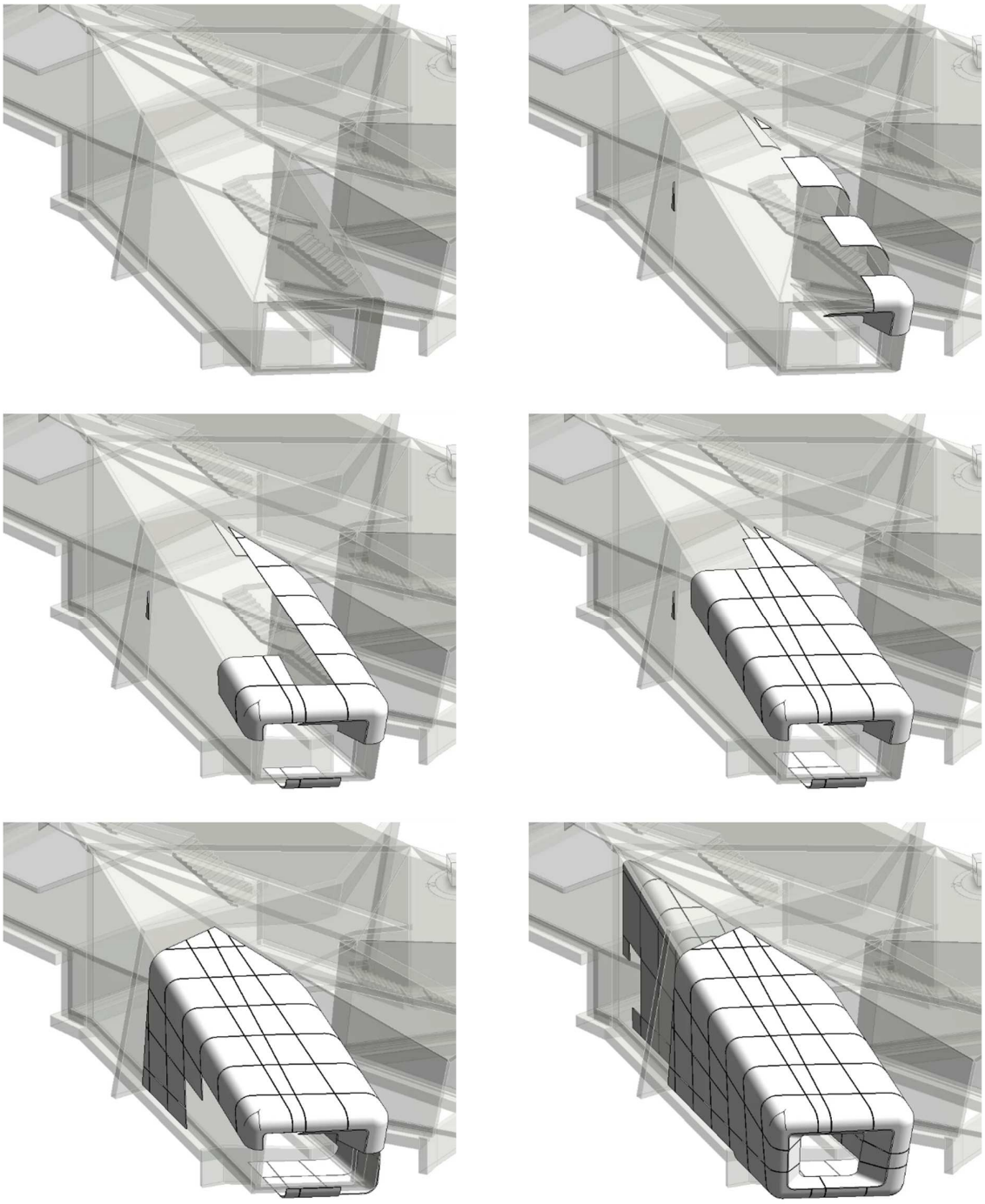


3. Select the script called “Family Panels by SAT.DYN”;
4. The script reads the exported SAT file and uses the node called “**MW FamilyInstance.ByGeometry**” from the **Springs Nodes** (https://dynamonodes.com/category/springnodes/) package to create single families by specifying the category, the RFT family template path and the families name;

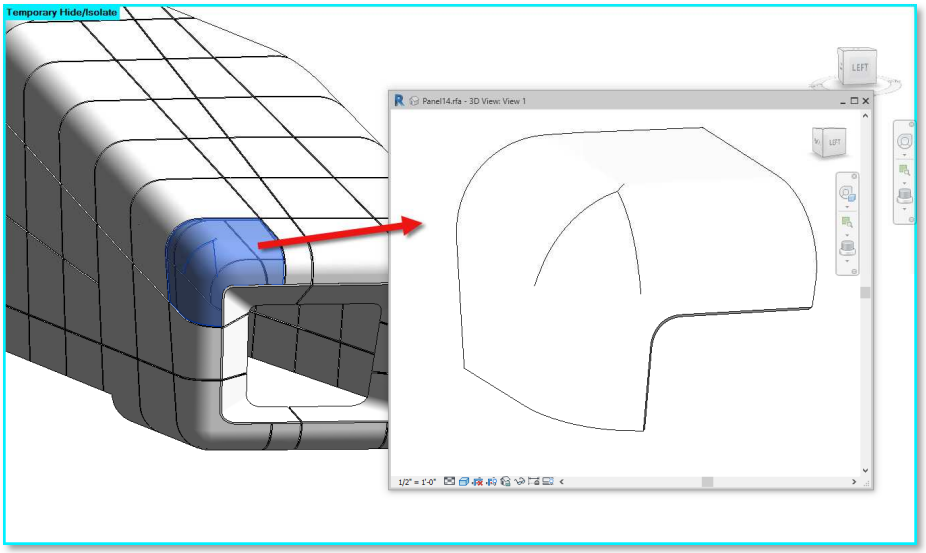


5. Click on “Run” when ready;
6. The script uses the family template specified to create a family with each solid in the SAT model;
7. This node also allows specifying the material to be assigned to the family geometries if this is defined inside the indicated family template;

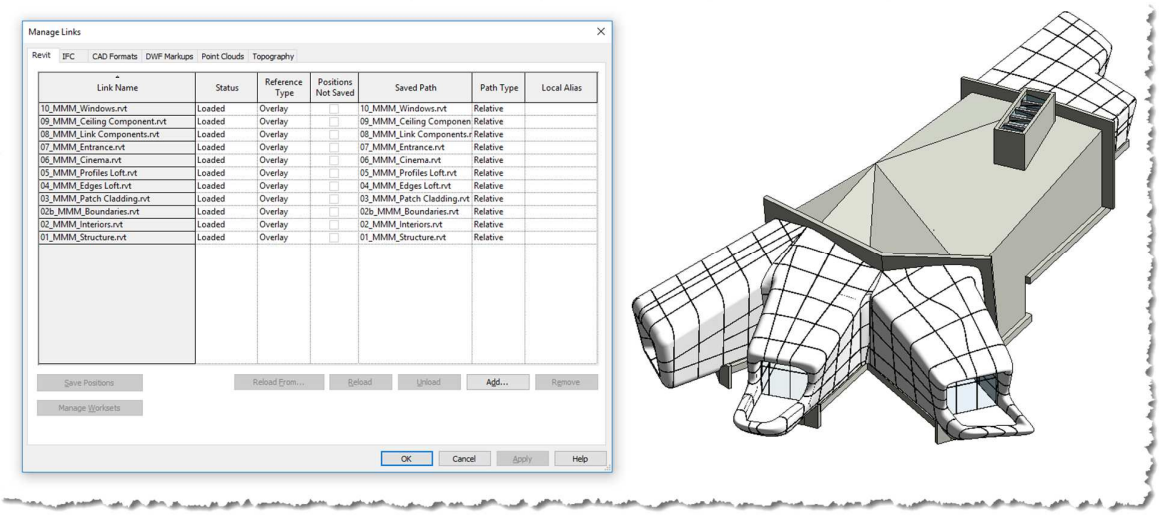
8. The script execution takes up to some minutes depending on the quantity and complexity of the SAT file to be “translated”;
9. Below images show some steps of the translation process;



10. The families can be individually opened and edited and belong to the “Casework” category;

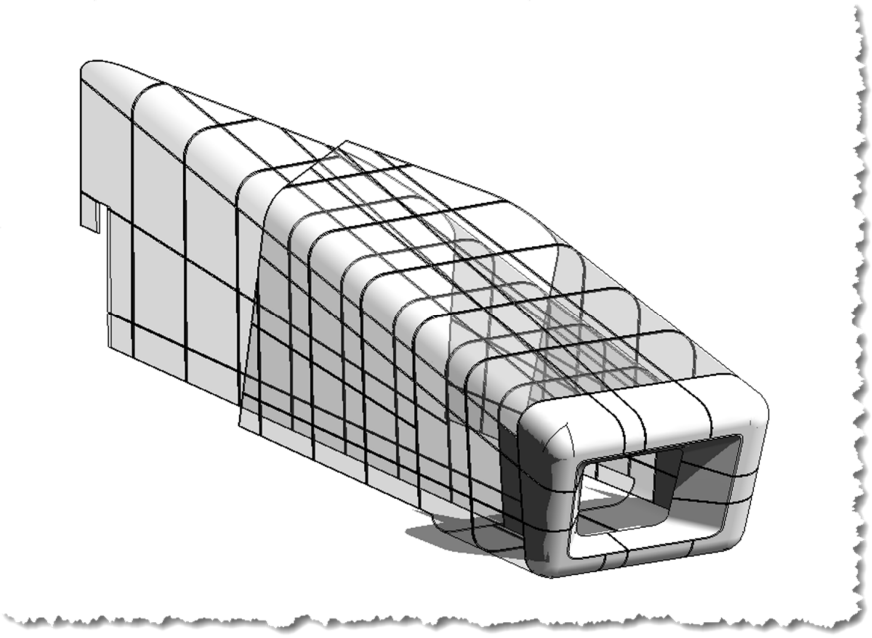


11. Create a file for each Fusion 360 component and link them to create an aggregate model;

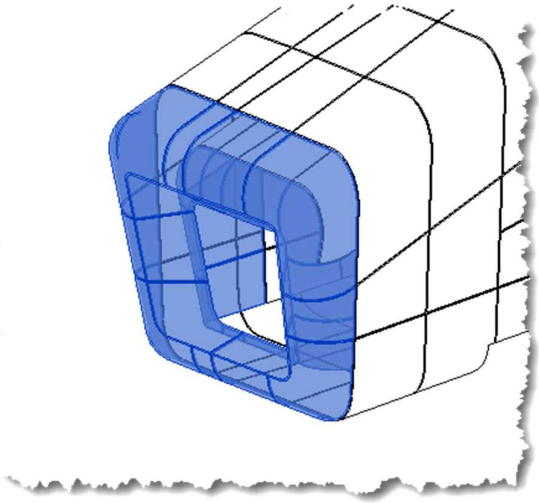


15 Exercise #7: Drawing Production for Panels Assemblies

This exercise shows how to use Revit to define panels assemblies and how to produce the related drawings.

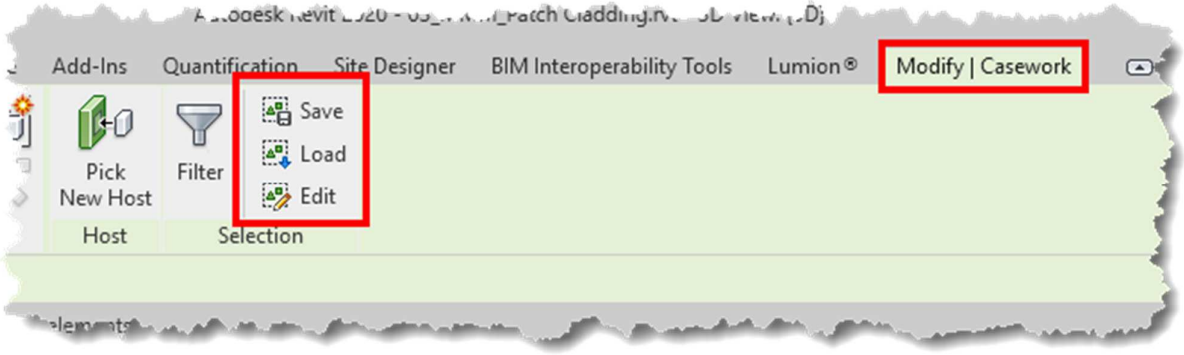


1. Open the file “03_MMM_Patch Cladding.RVT”;
2. Activate a 3D view and select the panels indicated on the next image;

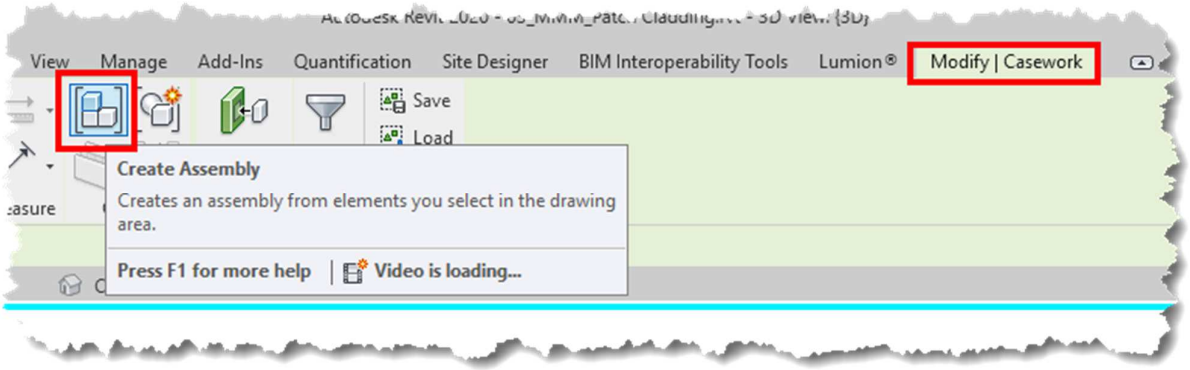


3.

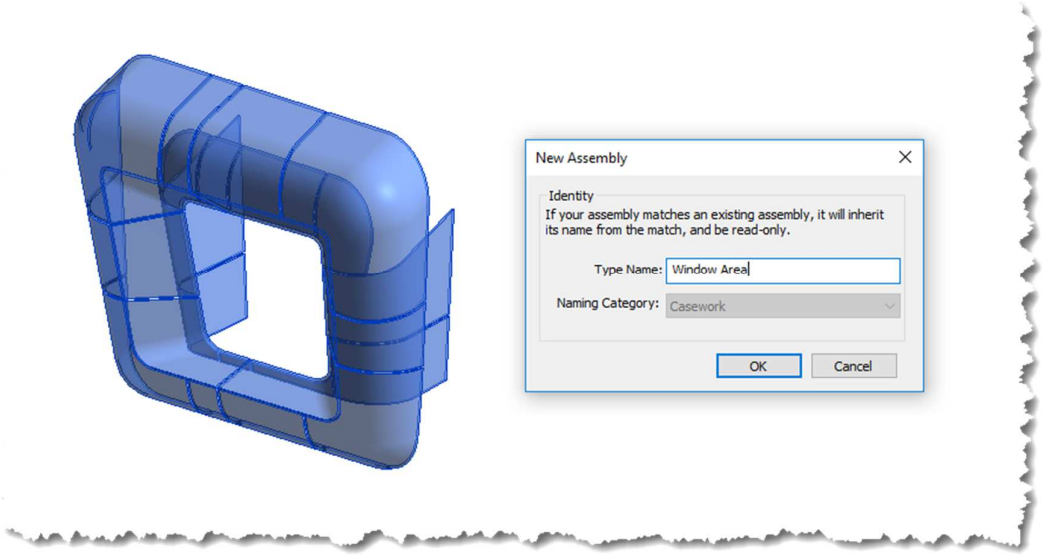
3. It is possible to save the selection set to recall the same selection when needed. After elements have been selected, click on “Save” under the contextual “Modify | Casework” ribbon: a dialog appears and requires specifying a name;



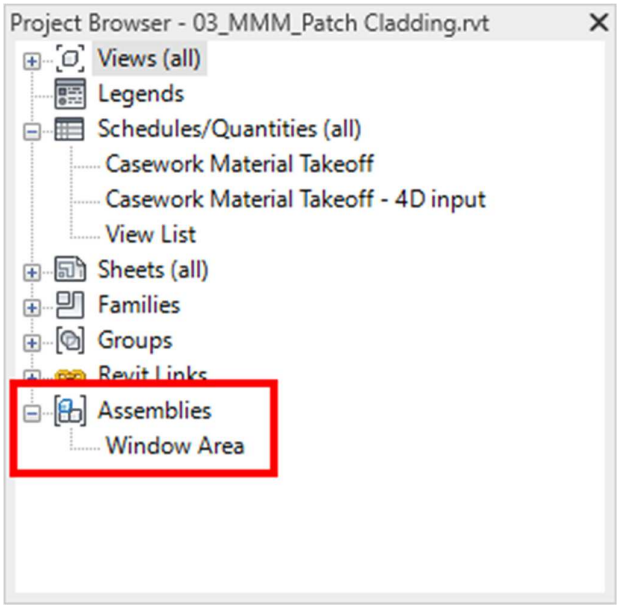
4. Once the elements are selected, click on the “Create Assembly” command;



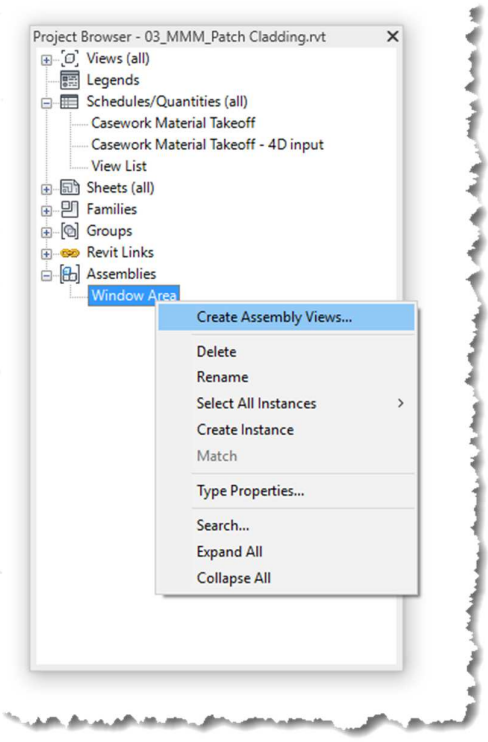
5. A dialog appears in which the assembly name must be specified. Name “Window Area” the new assembly;



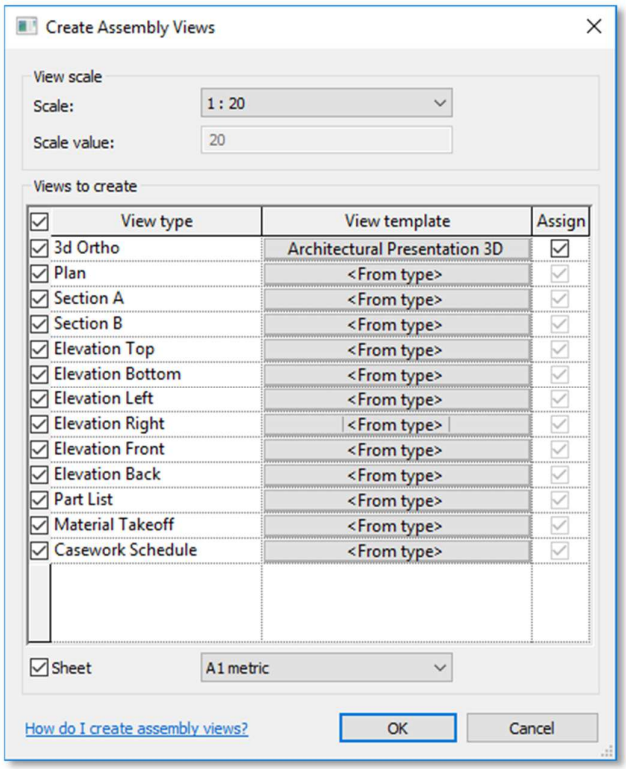
6. Once the assembly has been created, this appears in the Project Browser under the “Assemblies” group;



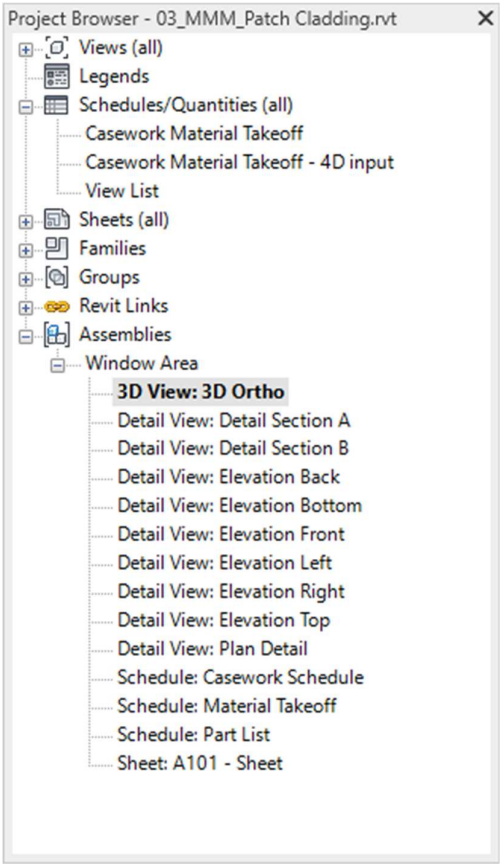
7. Right click on the “Windows Area” assembly and select “Create Assembly Views…”;



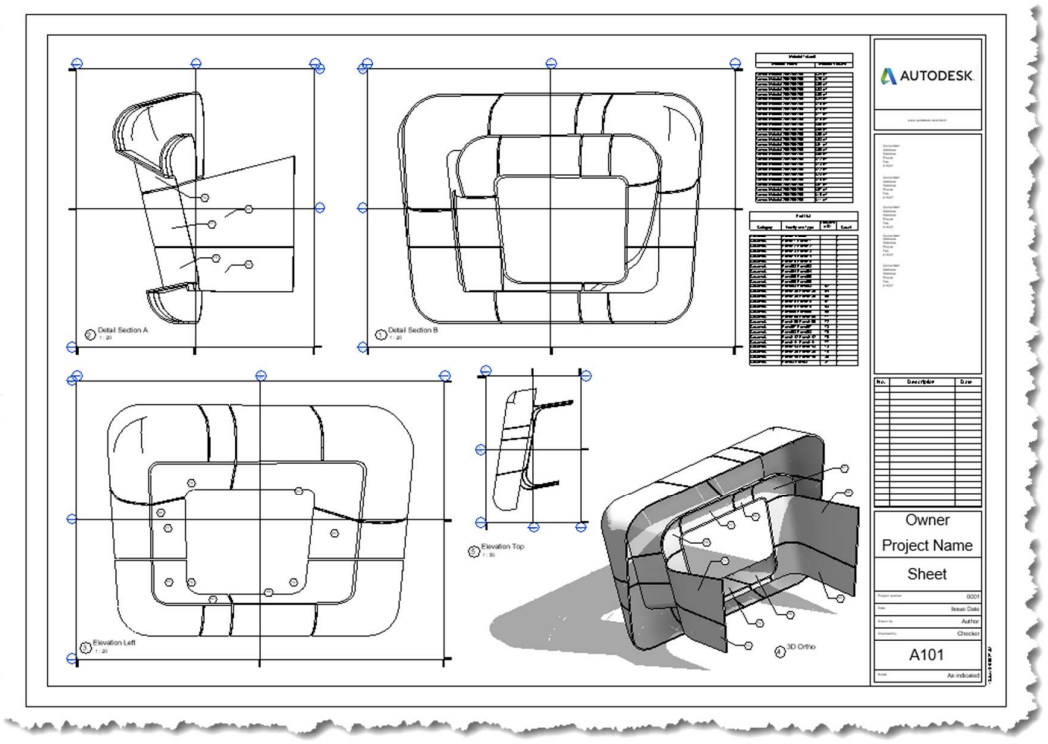
8. A dialog appears in which it is possible to choose which view must be created and the related View Template to be used. It also allows to create a drawing Sheet;



9. The created views are now listed below the “Window Area” assembly and if needed, they can be opened and modified;



10. Double click on the Sheet to open it. Once the Sheet is active, it is possible to place the relevant views by using the drag & drop method. In this case, panels are tagged with a Caseworks tag reporting the ID Sequence (shared parameter);



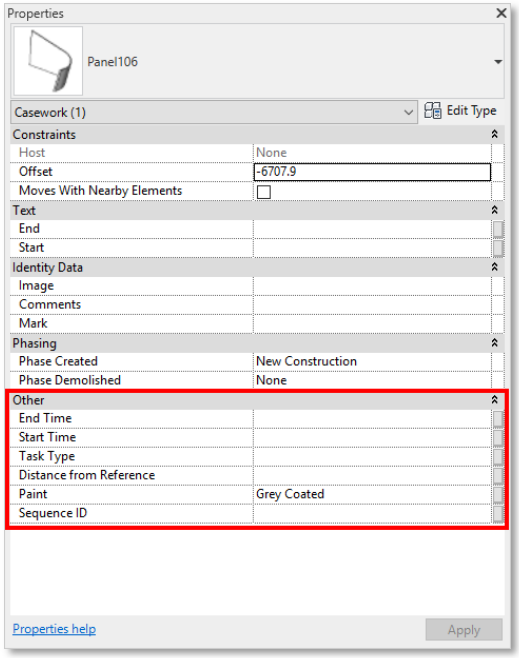
16 Exercise #8: 4D Sequencing for Panels Installation

The scope of this exercise is preparing the input for the panels' installation sequencing. In this simulation this happens from the internal beginning of the branch outwards.

The following image (https://www.floornature.com/messner-mountain-museumzaha-hadid-architects-10783/) shows the actual panel installation on the Messner Mountain Museum. Their placement begins from the interior of the building and this exercise replicates the same behavior.

16.1 Preparing the 4D input

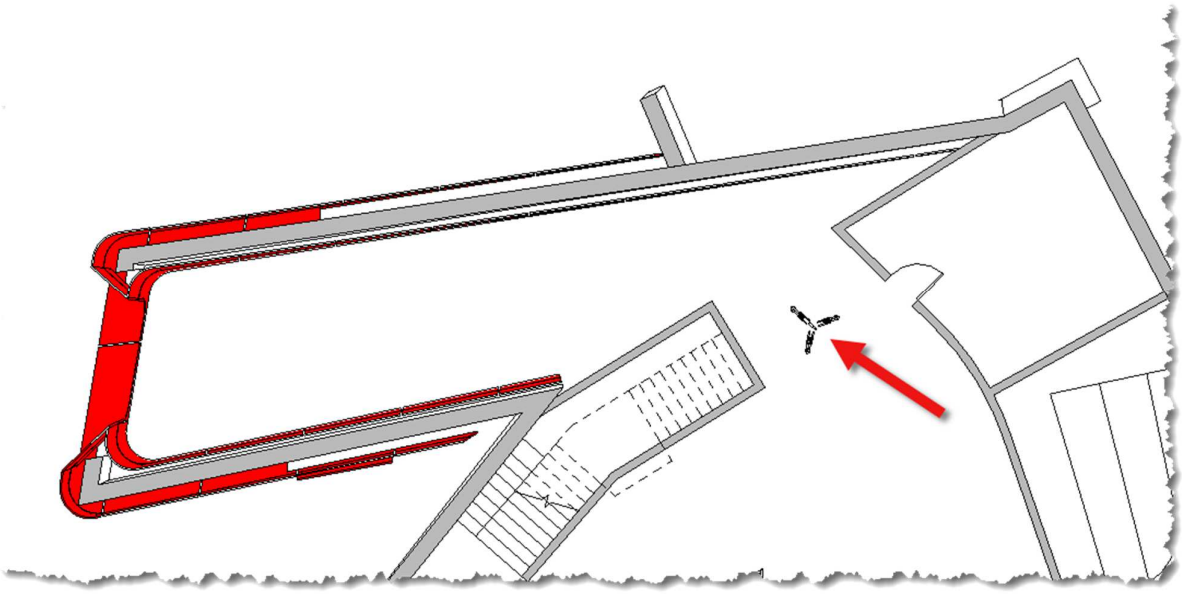
Open the "03_MMM_Patch Cladding.RVT" file. This has been setup with some parameters to specify the installation order for each cladding panel as reported in the previous part of this document. Customized parameters are shown in the picture below and they are related to the distance from the reference (the Theodolite placeholder) and their Sequence Id, which is the distance order form the reference.



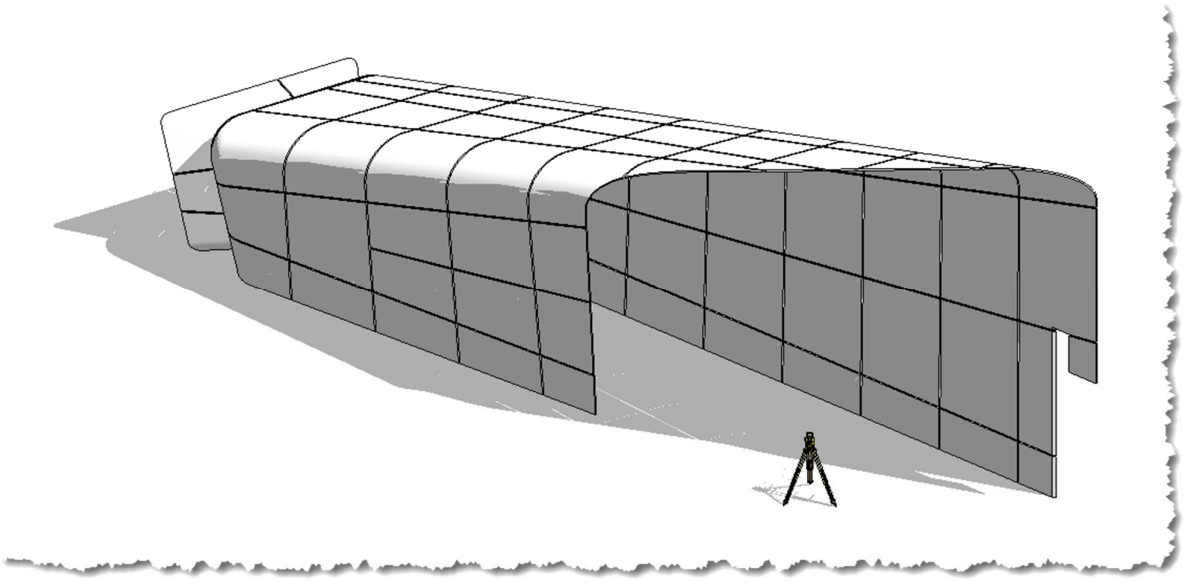
There are also parameters related to the installation timing: start and end time are assigned using the dynamo script in which also the installation needed time for each panel is specified.

In this case a specific family is used to define the origin for the installation sequence; this has been modeled using the shape of a survey theodolite.

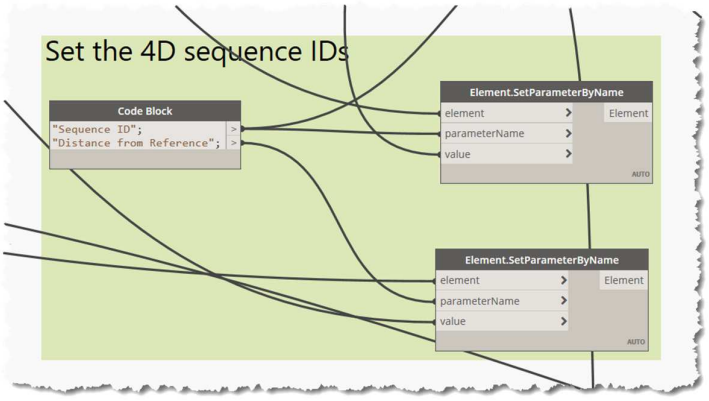
1. Place the family called “Theodolite → Type 1” as indicated in the next picture;



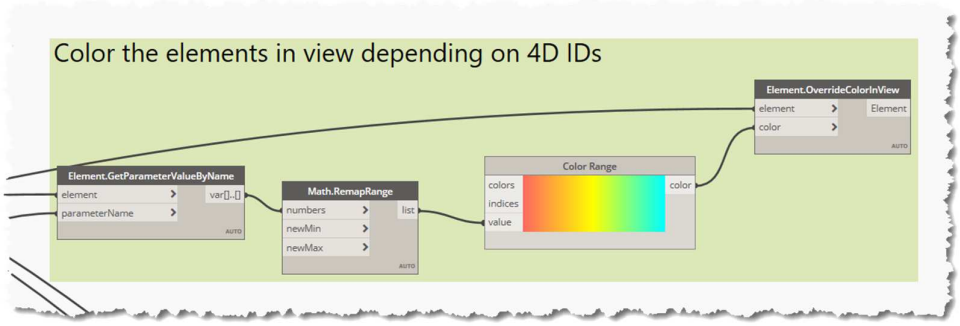
2. Start Dynamo 2.02 (or more recent versions) from the “Manage” ribbon;
3. Open the “Panels – Sequence ID.DYN” script;
4. The first part of the script, shown later, requires selecting the reference element (the Theodolite) and the panels to be numbered. For This reason, open a 3D view and isolate the panels to be used for the sequencing analysis. Use the next image as a reference;



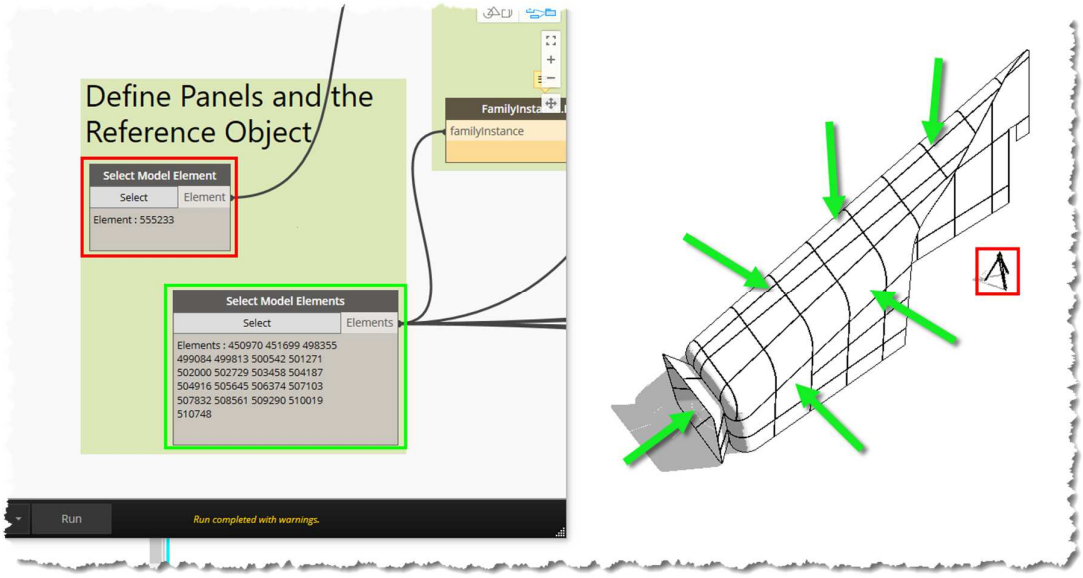
5. The dynamo script measures the distance of each panel form the placeholder (the Theodolite family) and populates the panels' attributes with the detected distance and their order from the closer to the farther panel;



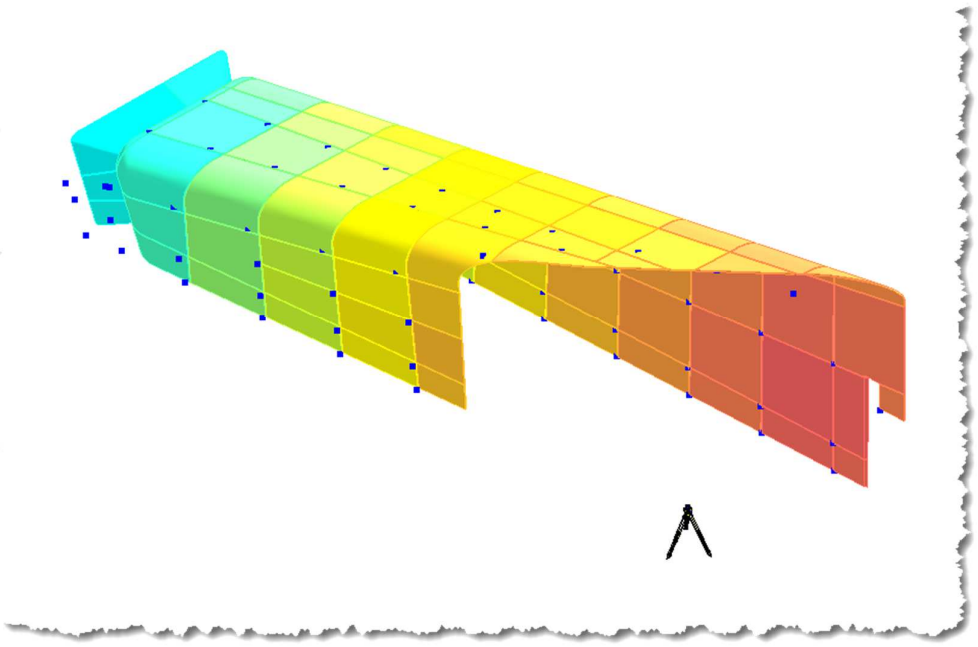
6. The script also applies a color to panels depending on their distance from the reference element. This color is applied to the current view only;



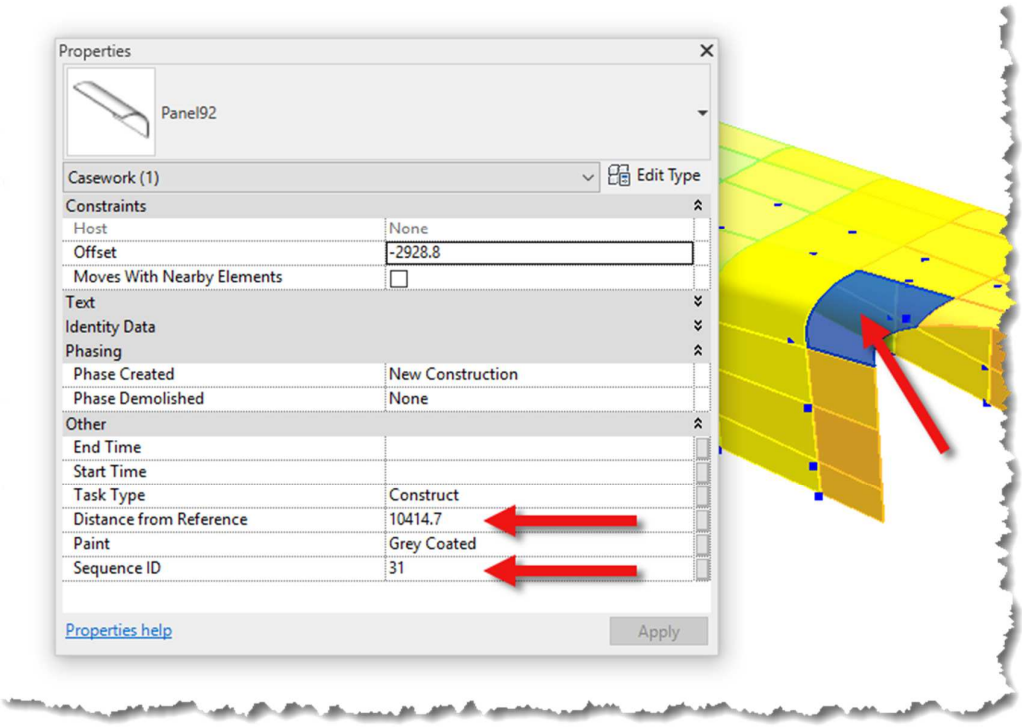
7. The script requires an input as shown in the next image: select the reference (red box and arrows) and then the set of panels to be analyzed (green indications);



8. Click “Run” to execute the script; all selected panels will acquire a color in the current view depending on their relative position against the Theodolite placeholder: red for the closer and cyan for the farther;



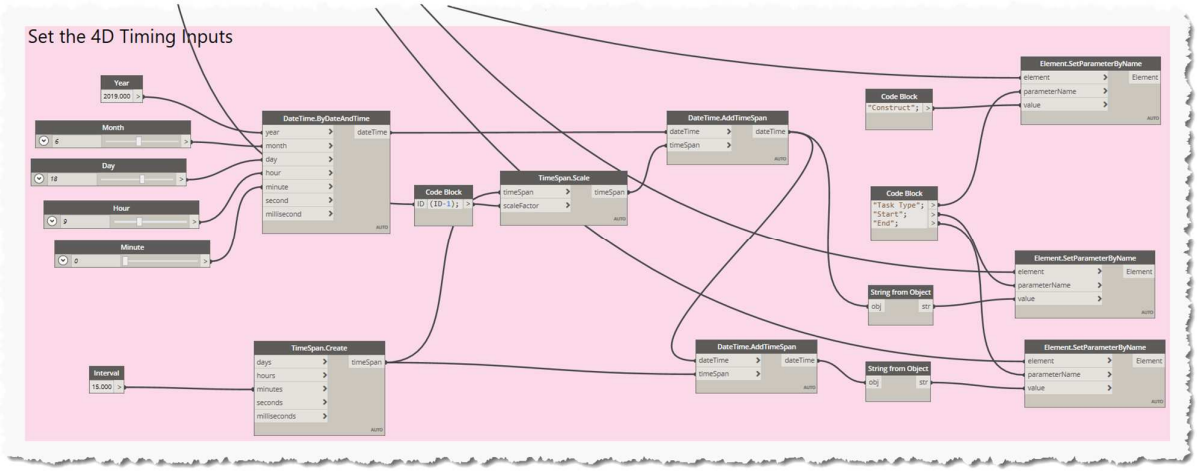
9. For instance: the panel shown below is the 31st panel to be installed and has a distance of about 10.4 meters from the reference element;



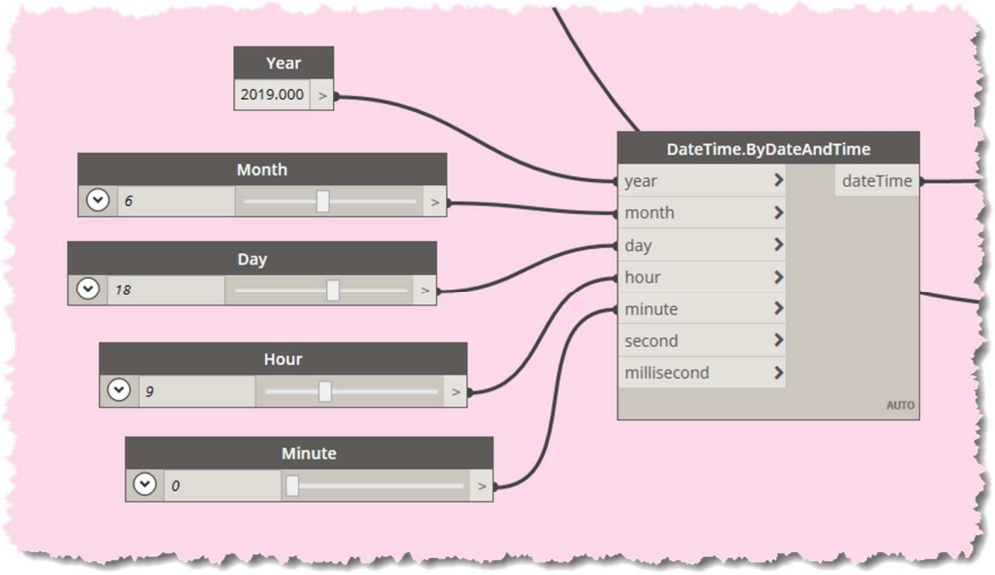
16.2 The 4D sequencing in Navisworks

The Dynamo script also writes the installation timing (both start and end) inside each panel. Revit is used to create a schedule to list each panel and the related timings so that this becomes an input to simulate the installation sequence in Navisworks Manage.

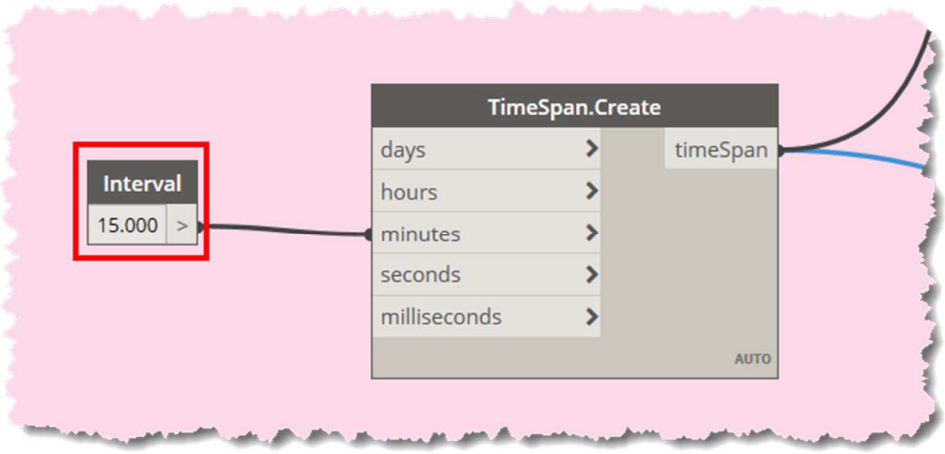
Thes script portion (shown in the next picture) allows to specify the start time for the installation activity and the duration of each panel placement. Then writes inside each panel the start and end time.



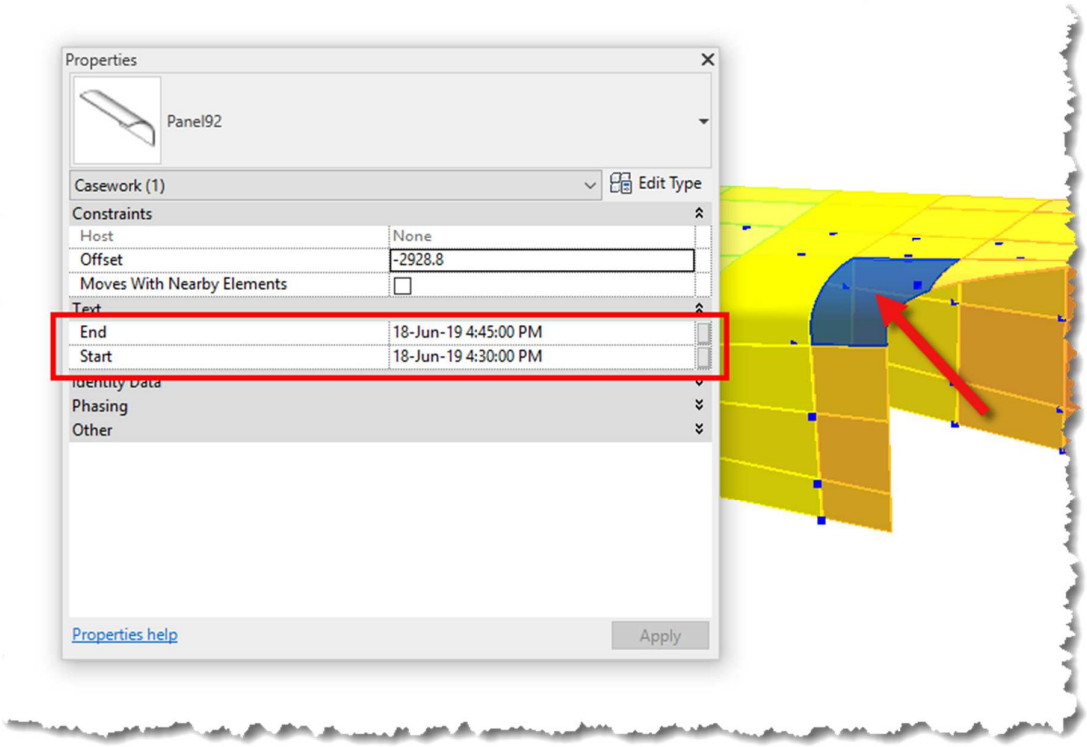
1. Specify the installation start defining: year, month, day, hour and minute;



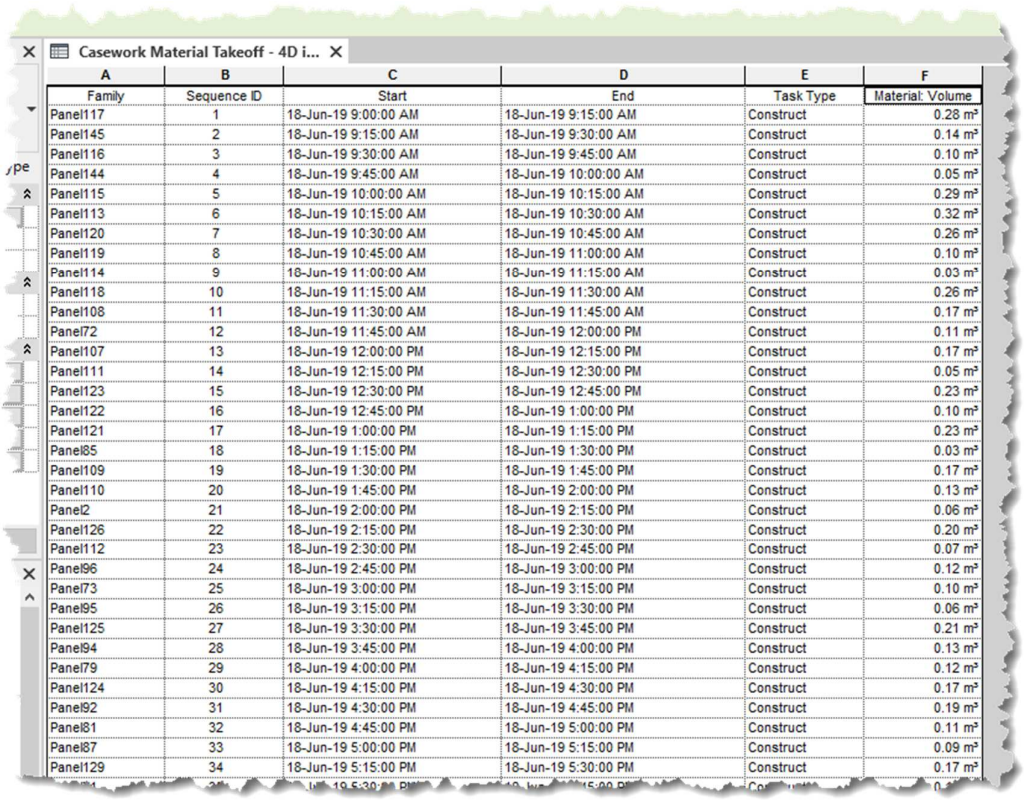
2. Define the time needed to install the single panel;



3. The following image shows how the same panel taken as a sample before, also reports the timing for the installation activity;



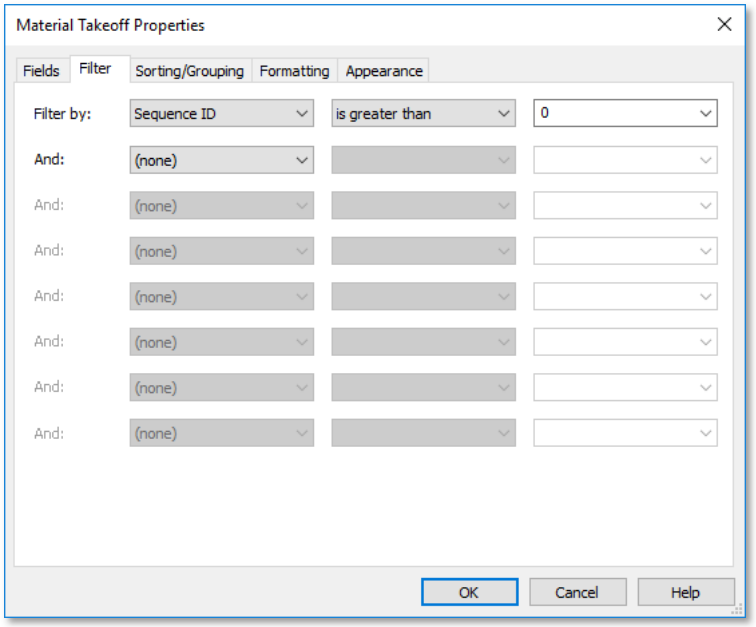
4. A schedule can be created to list all relevant information to perform a 4D sequencing simulation;



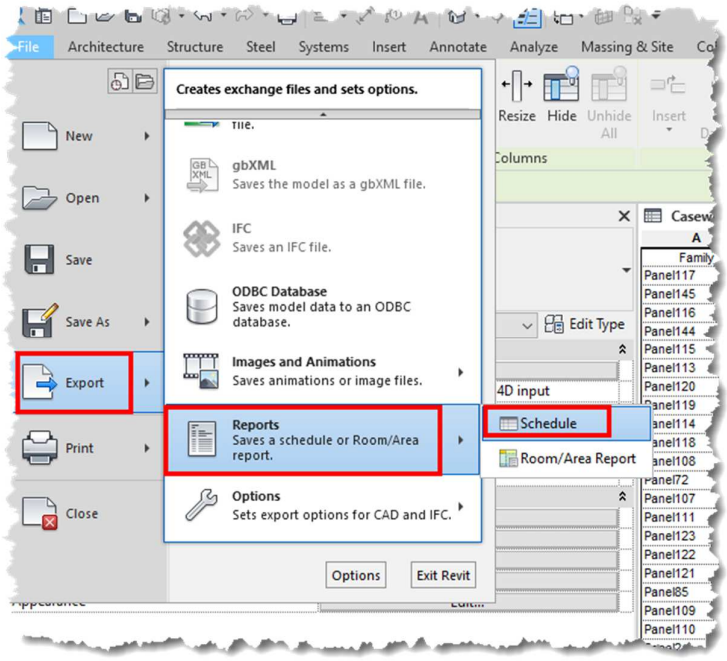
Casework Material Takeoff - 4D i...

A	B	C	D	E	F
Family	Sequence ID	Start	End	Task Type	Material: Volume
Panel117	1	18-Jun-19 9:00:00 AM	18-Jun-19 9:15:00 AM	Construct	0.28 m³
Panel145	2	18-Jun-19 9:15:00 AM	18-Jun-19 9:30:00 AM	Construct	0.14 m³
Panel116	3	18-Jun-19 9:30:00 AM	18-Jun-19 9:45:00 AM	Construct	0.10 m³
Panel144	4	18-Jun-19 9:45:00 AM	18-Jun-19 10:00:00 AM	Construct	0.05 m³
Panel115	5	18-Jun-19 10:00:00 AM	18-Jun-19 10:15:00 AM	Construct	0.29 m³
Panel113	6	18-Jun-19 10:15:00 AM	18-Jun-19 10:30:00 AM	Construct	0.32 m³
Panel120	7	18-Jun-19 10:30:00 AM	18-Jun-19 10:45:00 AM	Construct	0.26 m³
Panel119	8	18-Jun-19 10:45:00 AM	18-Jun-19 11:00:00 AM	Construct	0.10 m³
Panel114	9	18-Jun-19 11:00:00 AM	18-Jun-19 11:15:00 AM	Construct	0.03 m³
Panel118	10	18-Jun-19 11:15:00 AM	18-Jun-19 11:30:00 AM	Construct	0.26 m³
Panel108	11	18-Jun-19 11:30:00 AM	18-Jun-19 11:45:00 AM	Construct	0.17 m³
Panel72	12	18-Jun-19 11:45:00 AM	18-Jun-19 12:00:00 PM	Construct	0.11 m³
Panel107	13	18-Jun-19 12:00:00 PM	18-Jun-19 12:15:00 PM	Construct	0.17 m³
Panel111	14	18-Jun-19 12:15:00 PM	18-Jun-19 12:30:00 PM	Construct	0.05 m³
Panel123	15	18-Jun-19 12:30:00 PM	18-Jun-19 12:45:00 PM	Construct	0.23 m³
Panel122	16	18-Jun-19 12:45:00 PM	18-Jun-19 1:00:00 PM	Construct	0.10 m³
Panel121	17	18-Jun-19 1:00:00 PM	18-Jun-19 1:15:00 PM	Construct	0.23 m³
Panel85	18	18-Jun-19 1:15:00 PM	18-Jun-19 1:30:00 PM	Construct	0.03 m³
Panel109	19	18-Jun-19 1:30:00 PM	18-Jun-19 1:45:00 PM	Construct	0.17 m³
Panel110	20	18-Jun-19 1:45:00 PM	18-Jun-19 2:00:00 PM	Construct	0.13 m³
Panel2	21	18-Jun-19 2:00:00 PM	18-Jun-19 2:15:00 PM	Construct	0.06 m³
Panel126	22	18-Jun-19 2:15:00 PM	18-Jun-19 2:30:00 PM	Construct	0.20 m³
Panel112	23	18-Jun-19 2:30:00 PM	18-Jun-19 2:45:00 PM	Construct	0.07 m³
Panel96	24	18-Jun-19 2:45:00 PM	18-Jun-19 3:00:00 PM	Construct	0.12 m³
Panel73	25	18-Jun-19 3:00:00 PM	18-Jun-19 3:15:00 PM	Construct	0.10 m³
Panel95	26	18-Jun-19 3:15:00 PM	18-Jun-19 3:30:00 PM	Construct	0.06 m³
Panel125	27	18-Jun-19 3:30:00 PM	18-Jun-19 3:45:00 PM	Construct	0.21 m³
Panel94	28	18-Jun-19 3:45:00 PM	18-Jun-19 4:00:00 PM	Construct	0.13 m³
Panel79	29	18-Jun-19 4:00:00 PM	18-Jun-19 4:15:00 PM	Construct	0.12 m³
Panel124	30	18-Jun-19 4:15:00 PM	18-Jun-19 4:30:00 PM	Construct	0.17 m³
Panel92	31	18-Jun-19 4:30:00 PM	18-Jun-19 4:45:00 PM	Construct	0.19 m³
Panel81	32	18-Jun-19 4:45:00 PM	18-Jun-19 5:00:00 PM	Construct	0.11 m³
Panel87	33	18-Jun-19 5:00:00 PM	18-Jun-19 5:15:00 PM	Construct	0.09 m³
Panel129	34	18-Jun-19 5:15:00 PM	18-Jun-19 5:30:00 PM	Construct	0.17 m³

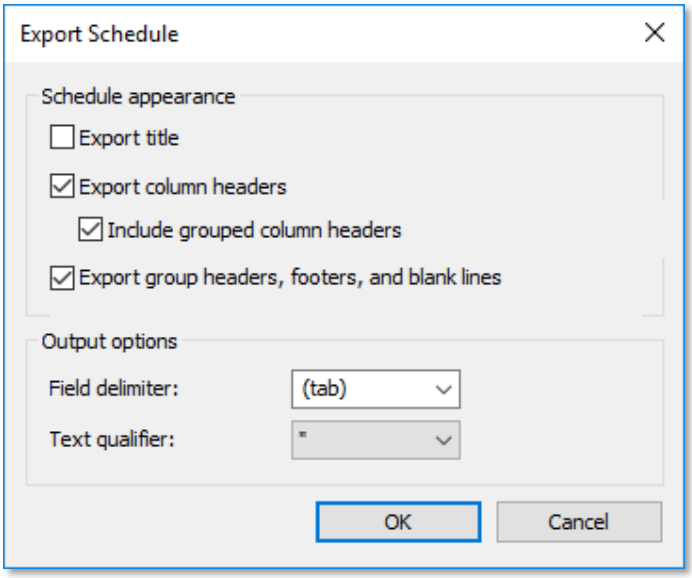
11. In this schedule a filter removes the elements not taken into consideration for the sequencing (for instance, in this case, the outer shell objects);



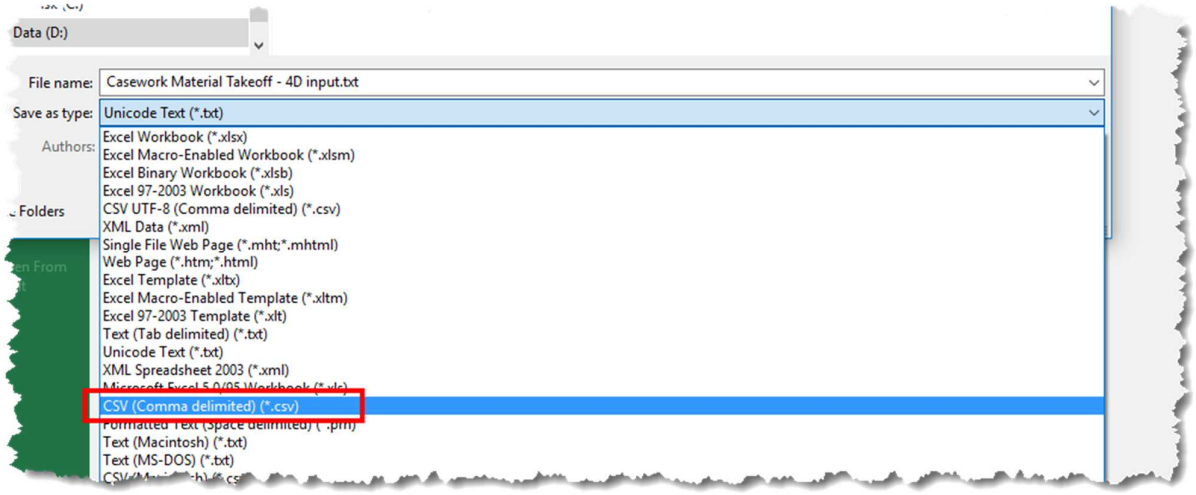
5. Export the schedule;



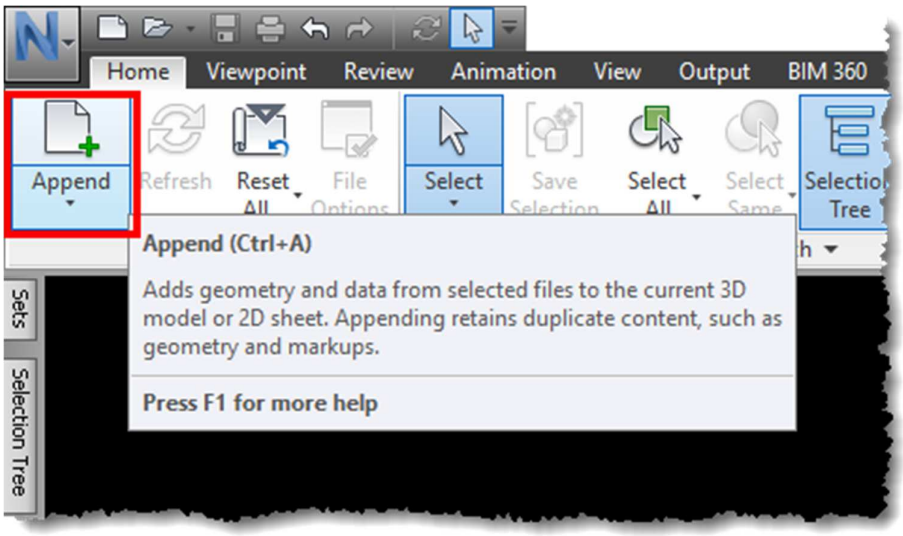
6. Configure the export setting as shown below. “Export Tile” should be unchecked although in this case the schedule title has been hidden in the same Revit schedule;



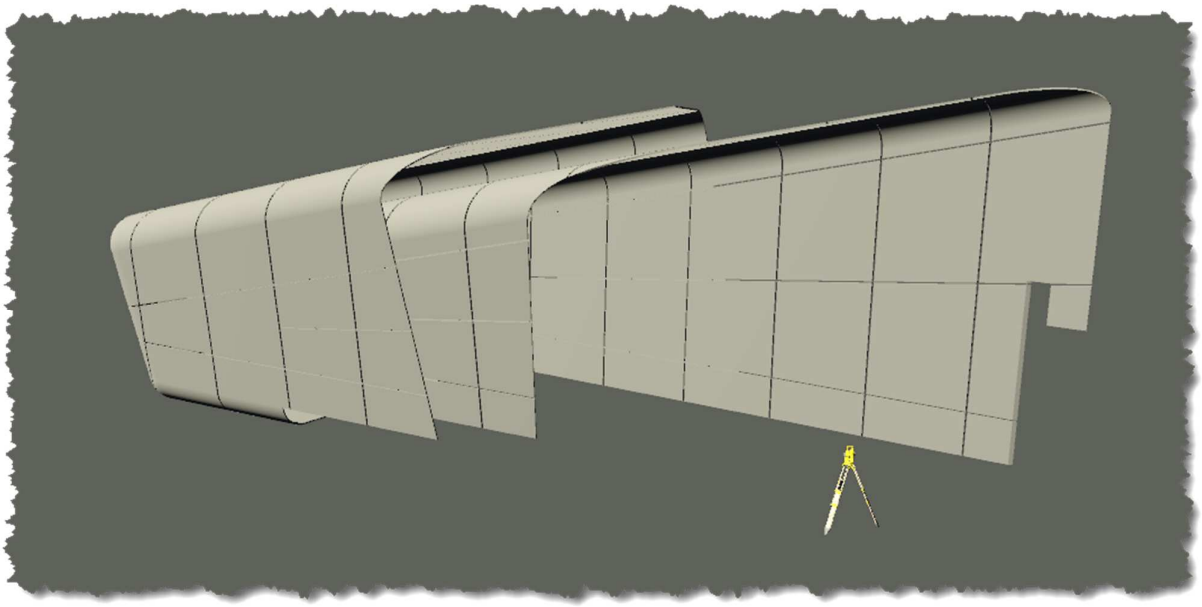
7. The produced file will be prepared using MS Excel. The file needs to be formatted so that Autodesk Navisworks Manage can use it; this must be a CSV with fields properly separated by commas. In order to maximize the compatibility, MS Excel will be used although the file could be exported with the “comma” field delimiter;
8. Select the created file and change its extension: from TXT to CSV. This is a workaround allowing MS Excel to directly open the file without passing through the standard import procedure;
9. Open the file using MS Excel and save it as CSV;



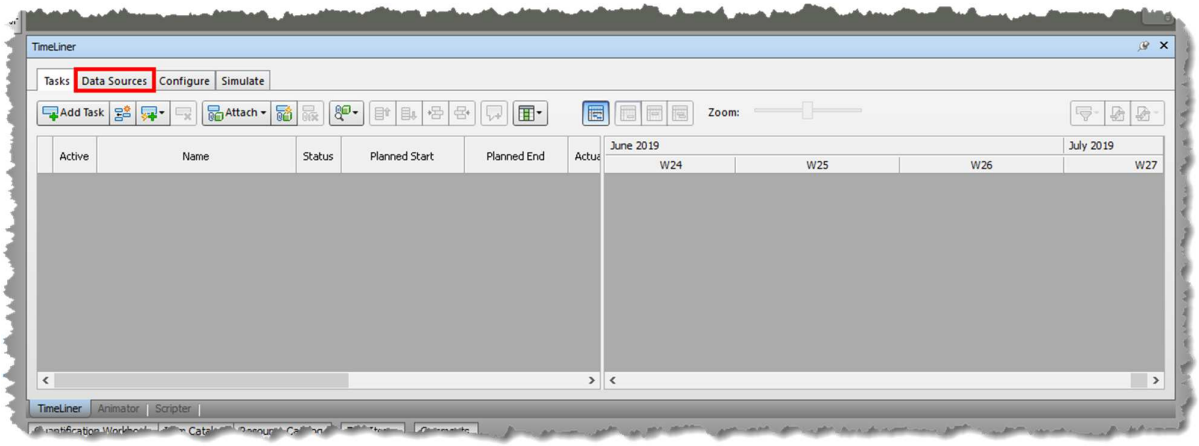
7. Open Autodesk Navisworks Manage;
8. Append the Revit file called “03_MMM_Patch Cladding.RVT”;



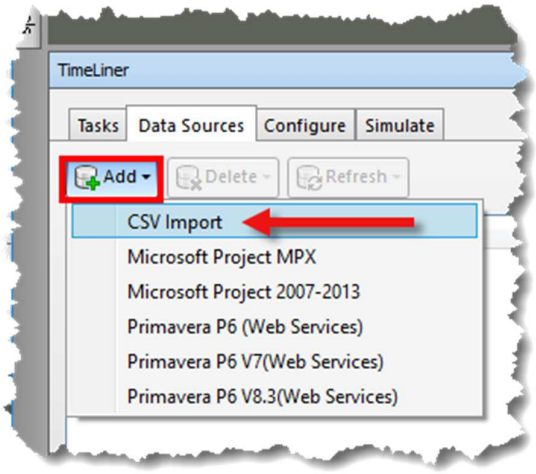
9. The cladding components appear on the work area of Navisworks;



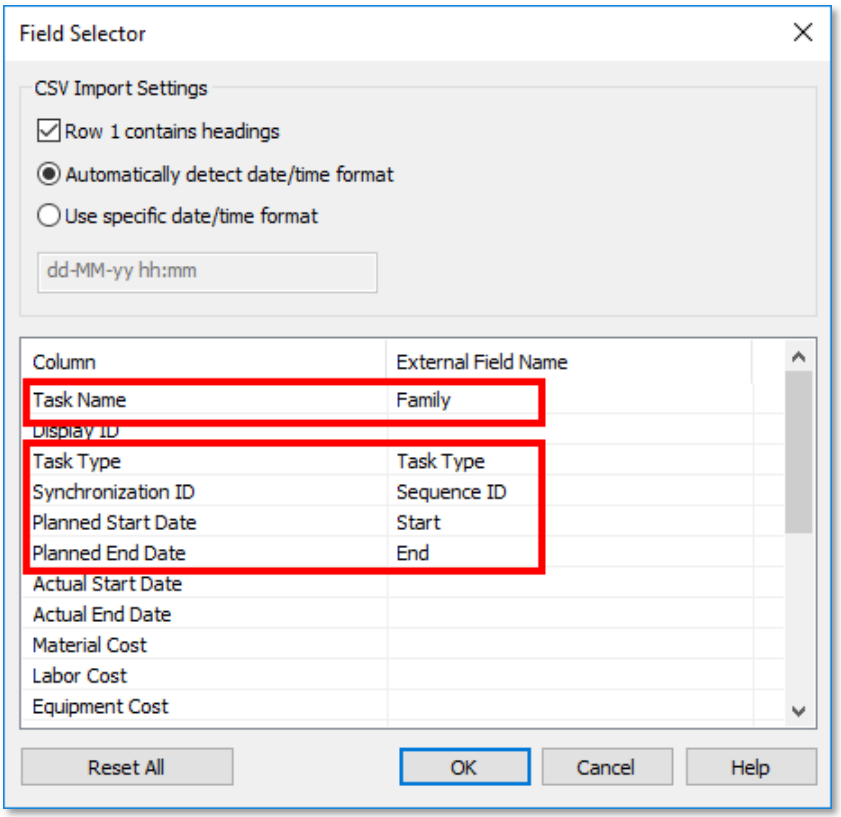
10. Open the “Timeliner” dialog and click on the “Data Sources” tab;



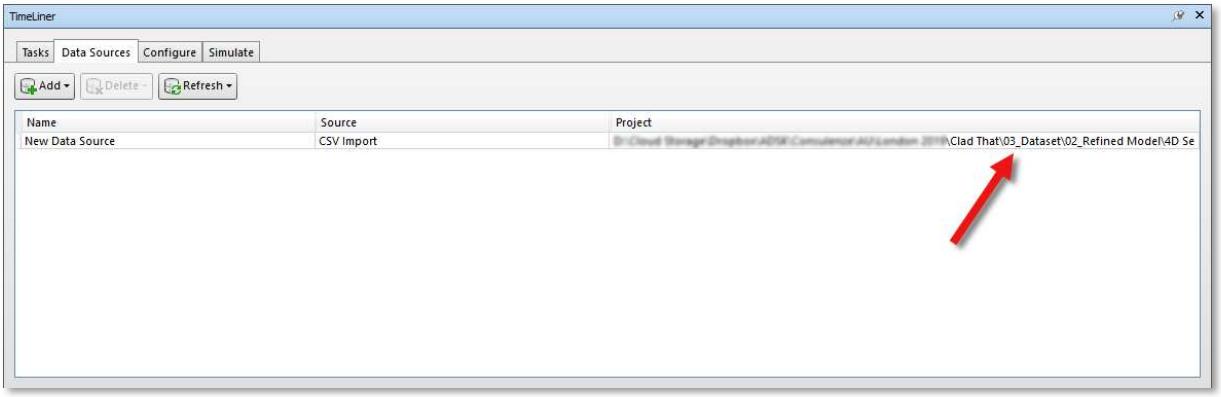
11. Click on “Add” and then select “CSV Import”;



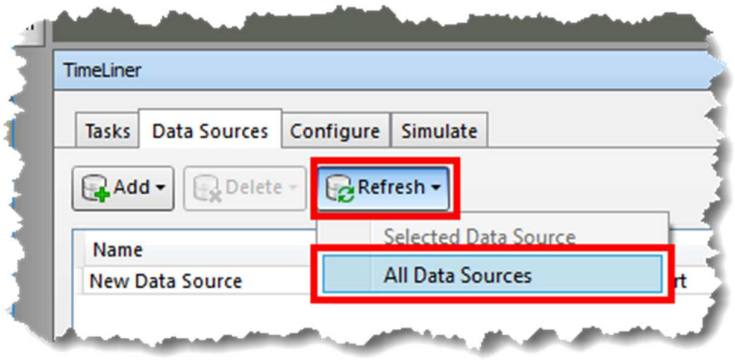
12. Select the file previously prepared with MS Excel, in this case this is called “4D_3_Casework Material Takeoff - 4D input.CSV”;
13. A dialog appears in which it is possible to match the columns in the CSV file with the 4D related fields; set the mapping as indicated in the next image and click “OK”;



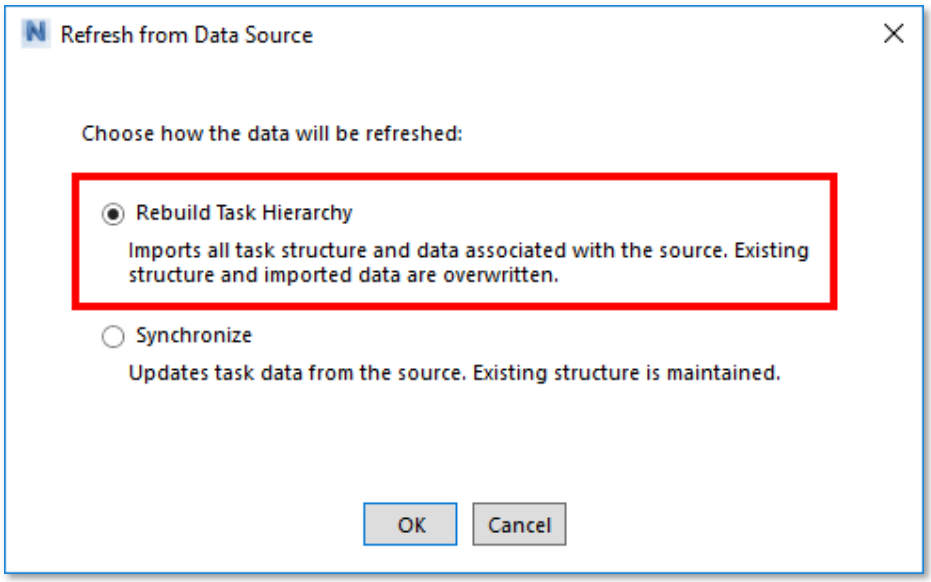
14. The file has been set as a data source and appears in the space below;



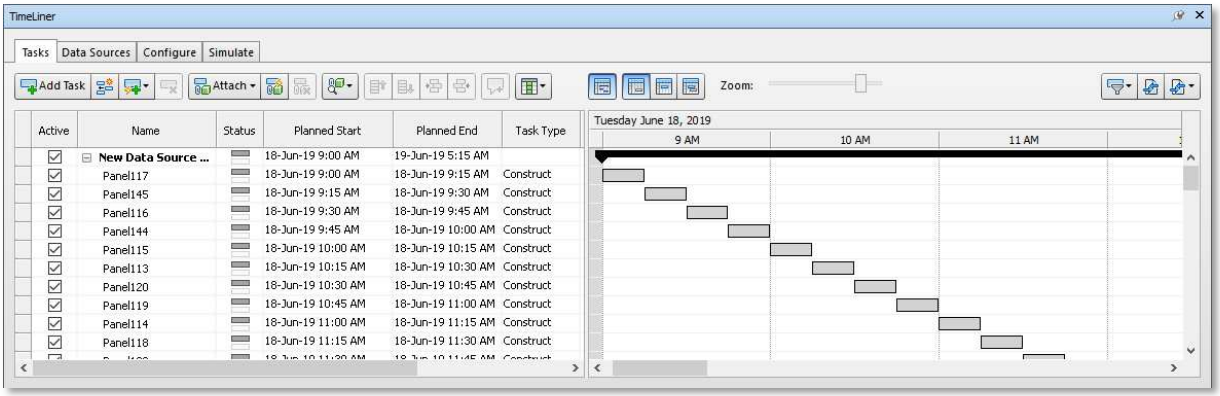
15. Click on the “Refresh” button and specify “All Data Sources”;



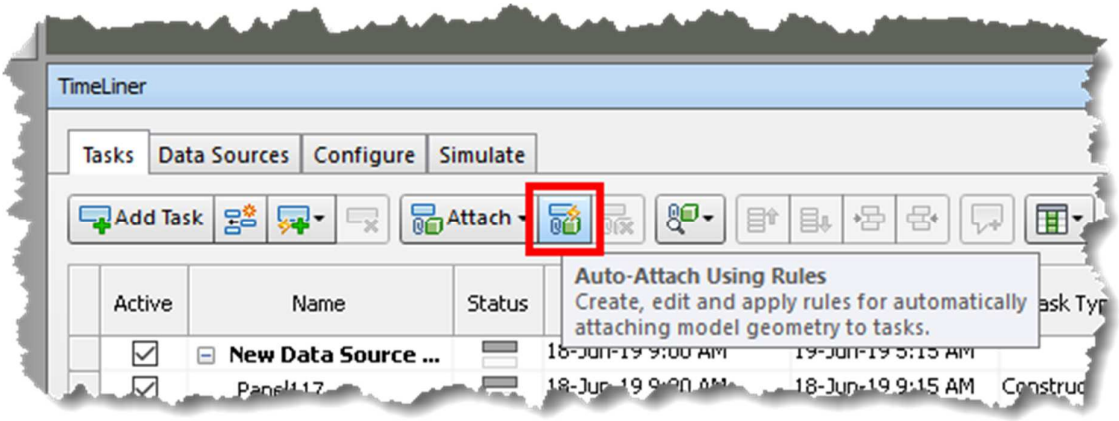
16. Being that this is the first time the CSV file is imported, specify “Rebuild Task Hierarchy” on the dialog that appears;



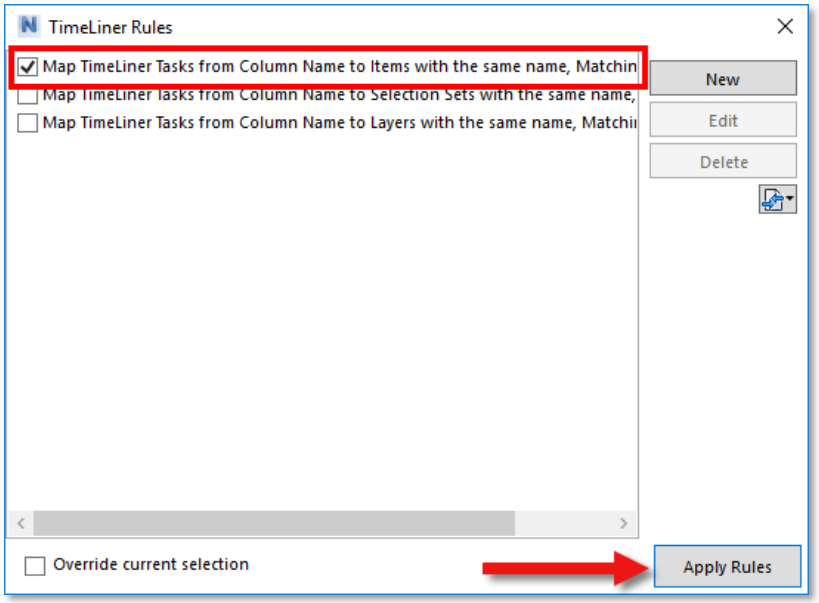
17. Navisworks created one task for each panel family / panel instance (there is a single instance for each family in the Revit model). Each task used the data provided with the Revit schedule to assign a start and an end time with the duration defined in the Dynamo script. Also, the Task type has been defined using Dynamo which wrote the specific information in Revit;



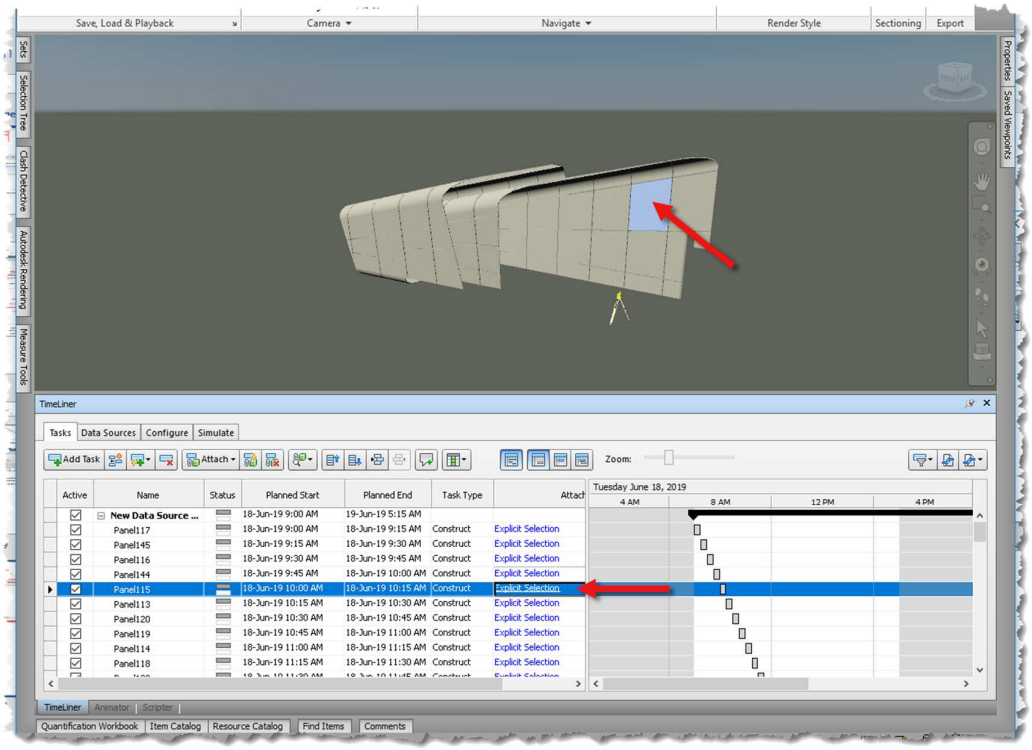
18. Select the command “Auto-Attach Using Rules”; this allows to automatically attach to each task a specific set of elements in the model;



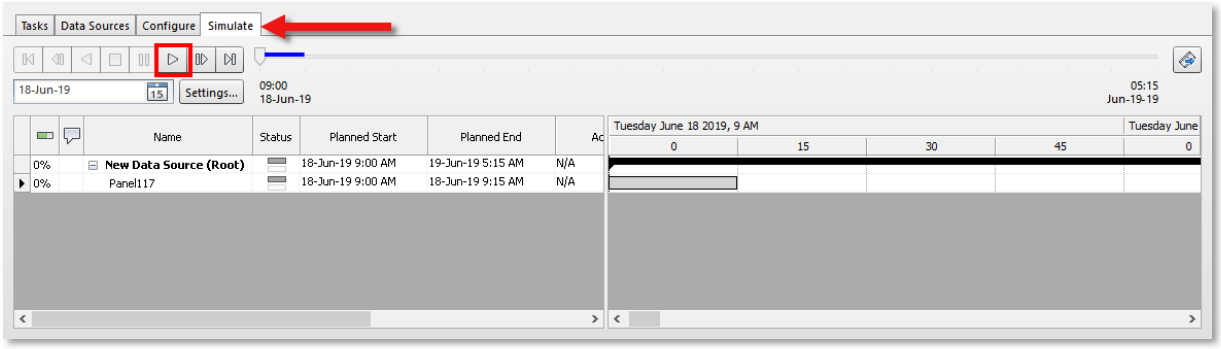
19. In this case, each task has the same name of the related element since the task takes that from a Revit schedule which lists the model families. This means that the first rule can be used to attach to each task the items having the same name. Click on “Apply Rules” and close the dialog;



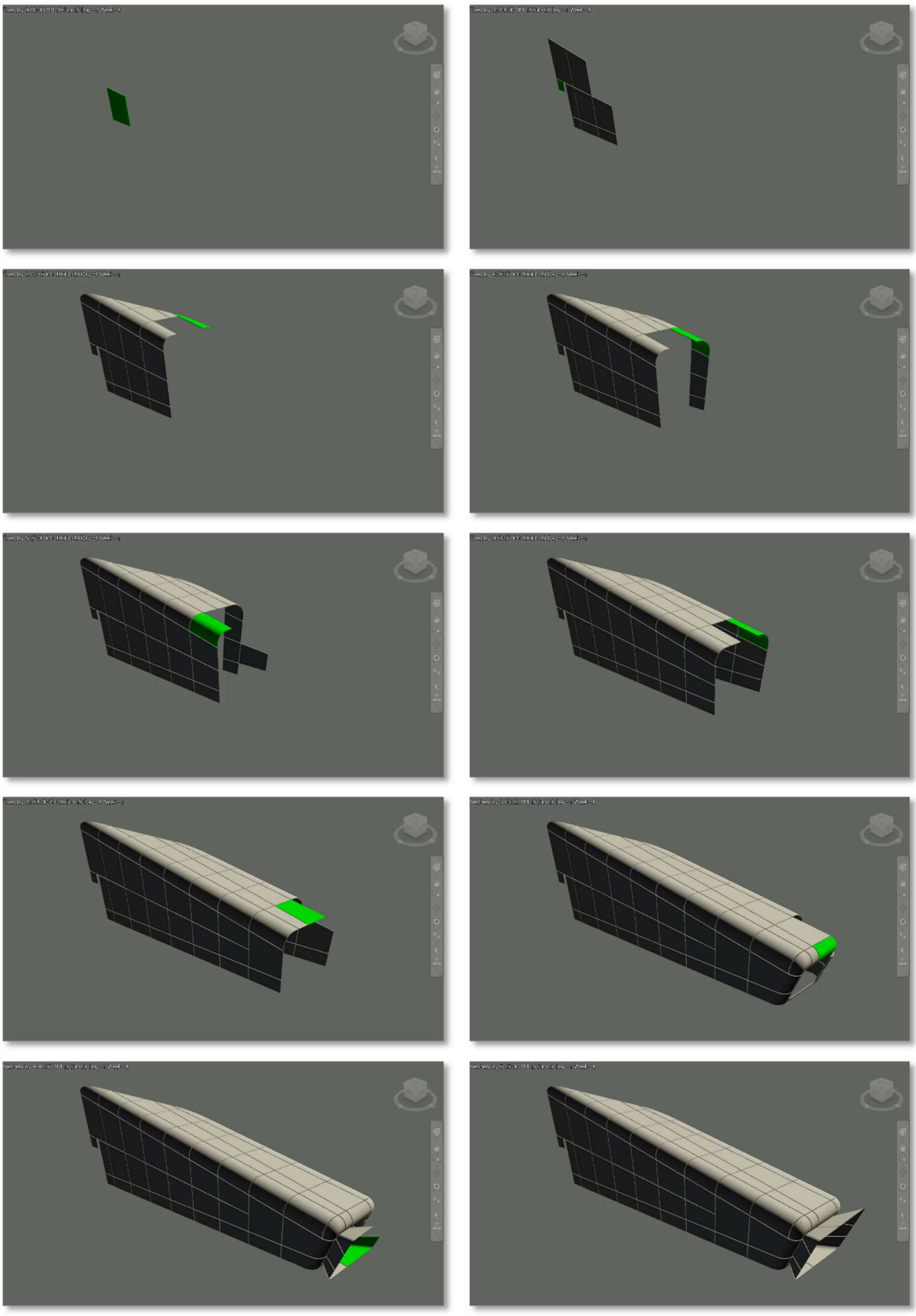
20. The Navisworks Timeliner shows how each task is directly connected with a single panel in the model;



21. Select the “Simulate” tab and click the “Play” symbol;



22. Navisworks shows an animation in which the panels are placed one by one, respecting the Gantt diagram created before; images on next page show frames extracted from the Timeliner simulation;



17 Appendix: Keyboard Shortcuts

Speed up your design work with keyboard shortcuts.

Use these shortcuts to streamline the interaction with the Fusion 360 software.

Animate Workspace Commands

shortcut	command
U	Auto Explode All Levels
E	Manual Explode
P	Publish Video
M	Transform Components
C	View

CAM Workspace Commands

shortcut	command
Ctrl + D | Command +D	Duplicate
Ctrl + G | Command + G	Generate Toolpath
Shift+S	Scripts and Add-Ins
Ctrl + L | Command + L	Show Log

Canvas Selection

Shortcut	command
Ctrl + C | Command + C	Copy
Ctrl + X | Command + X	Cut
Hold Shift + Hold Middle Mouse Button	Orbit
Hold Shift + Click then Hold Middle Mouse Button	Orbit around point
Hold Middle Mouse Button	Pan
Ctrl + V | Command + V	Paste
Ctrl + Y | Command + Y	Redo
Ctrl + Z | Command + Z	Undo
Roll Middle Mouse Button	Zoom

Drawing Workspace Commands

shortcut	command
B	Balloon
C	Center Mark
Delete	Delete
D	Dimension
M	Move
P	Projected View
T	Text

Edit Form Commands

shortcut	command
Alt + Drag	Add geometry
Alt + Control + Drag | Alt + Command + Drag	Add geometry and keep creases

Modeling Commands

shortcut	command
R	2-point Rectangle
A	Appearance
Shift + J	As-built Joint
C	Center Diameter Circle
Ctrl + B | Command + B	Compute All
E	Extrude
F	Fillet
2	Freeform Selection
H	Hole
J	Joint
L	Line
I	Measure
S	Model Toolbox
M	Move
X	Normal / Construction
O	Offset
3	Paint Selection
Q	Press Pull
P	Project
Shift + S	Scripts and Add-ins
D	Sketch Dimension
Shift + N	Toggle Component Color Cycling
V	Toggle Visibility
T	Trim
1	Window Selection

Render Workspace Commands

shortcut	command
A	Appearance

Sculpt Workspace Selection

shortcut	command
Shift + Up Arrow	Grow selection
Alt + N | Command + N	Invert selection
Alt + O | Control + O	Loop grow selection
Alt + P | Control + P	Loop selection
Alt + Right Arrow | Control + Command + Right Arrow	Next U
Alt + Up Arrow | Control + Command + Up Arrow	Next V
Alt + Left Arrow | Control + Command + Left Arrow	Previous U
Alt + Down Arrow | Control + Command + Down Arrow	Previous V
Alt + M | Command + M	Range selection
Alt + K | Control + K	Ring grow selection
Alt + L | Control + L	Ring selection
Alt + J | Control + J	Ring shrink selection
Double-click an edge	Select edge ring
Select two faces then double-click a third face	Select face ring
Shift + Down Arrow	Shrink selection
Ctrl + 1 | Control + 1	Toggle box mode
Ctrl + 2 | Control + 2	Toggle control frame mode
Ctrl + 3 | Control + 3	Toggle smooth mode

Simulate Workspace Selection

shortcut	command
Ctrl + D | Command + D	DOF View
F	Force
Ctrl + G | Command + G	Groups View
Ctrl + L | Command + L	Model View
N	New Simulation Study
Ctrl + R | Command + R	Results View
E	Settings
C	Structural Constraint
L	Structural Loads

System

shortcut	command
Ctrl + Shift + S | Command + Shift + S	Recovery Save
Ctrl + S | Command + S	Save Version

IMAGINE,

DESIGN,

MAKE

A better World