Unlocking the Power of the Shape Generator & Modelling with Meshes

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Learning Objectives

- Find and set physical material properties for 3D printable materials in the Shape Generator
- Create complete part designs automatically using the Shape Generator
- Use the generated mesh directly in an assembly and for 3D printing
- Tweak a solid model using the resulting mesh using Inventor’s mesh to surface tools

Description

In this class we will follow new workflows available to designers from the introduction of Inventor’s shape generator, mesh handling toolset and modern 3D printing technologies. We will start by looking at 3D printable materials and their physical properties. Using these material specs, required for proper design simulation, we will then configure the shape generator to automatically create an optimized shape for us based on sample design requirements. With the resulting mesh, the class will examine the ins and outs of two separate workflow paths; using the generated mesh directly for manufacture and use in an assembly, or modifying the part using Inventor’s new mesh to surface tools to tweak the design. Join this class for a close up look at the possibilities realized if we combine Inventor’s shape generator and mesh handling with modern 3D printing capabilities.

Your AU Expert(s)

David received his Bachelor of Industrial Design from Carleton University in Ottawa, Ontario, Canada and is currently a Senior Applications Expert for IMAGINiT Technologies. He provides design and data management consulting and teaches fundamental and advanced level courses for Autodesk manufacturing applications. David began work as a mechanical designer in telecommunications for in-building radio equipment and then as an industrial designer at Mitel Networks. In years prior to working for IMAGINiT David also worked in the office furniture industry as a product engineer where he not only designed new product but also helped develop CAD to CAM workflows for wood and fabric CNC machine automation.
Introduction

Autodesk has been working toward generative design technologies for a number of years and we have recently seen the introduction of this technology into Inventor. If you were here at AU in 2014 you may remember during the opening keynote, the promise of generative was the focus of not only Jeff Kowalski’s keynote address but also it was also a major theme that year, last year and this year also.

![Figure 1 AU 2014 Keynote Address](image)

It’s exciting to see this technology’s arrival in a form that we can work with to produce real physical results from. First released with Inventor 2016 R2 the Inventor Shape Generator is Autodesk’s first foray into generative design with Inventor. The Shape Generator, and generative design, as a whole is a simulation design tool and uses the Nastran simulation engine to perform its analysis. If you are at all familiar with using Inventor Professional’s stress analysis environment you will feel right at home with the Shape Generator.

In this class we will look at new design and manufacturing workflows that you can implement even at this early stage of generative design using Inventor 2017 and the Shape Generator. With the release of Inventor 2017 comes a refined Shape Generator and added tools to handle the mesh output returned to your Inventor part model.

3D Printing Material Properties

There is sometimes a clash between the Shape Generator and traditional manufacturing methods. The result you get from a shape generator simulation will rarely be suitable for standard machining or molding processes unless you are designing plate or sheet metal parts. Apart from thin-walled parts, the Shape Generator has the greatest potential benefit with 3D printing and other advanced manufacturing processes.
Generative design technology is still young, and 3D printing is maturing, but the material specifications for the 3D printing process remain in their infancy. There is very little hard data on the physical characteristics of printed materials such as Young’s Modulus, Tensile Strength, Yield Strength etc. This information is critical for proper design simulation. Even specifications that are available have to be looked at critically to see if the values are for the raw materials or for the finished product, a generalization or machine specific.
FIGURE 3 SOME SERVICES LIKE SHAPEWAYS POST THEIR MATERIAL DATA WHICH MAKES SIMULATION USING GENERATIVE DESIGN TOOLS FOR 3D PRINTING POSSIBLE.

So, while we can consider the physical properties of the raw materials used, the process must be considered as well since it will affect the final state of the material. Differences occur by raw material, of course, but also by process and even machine to machine within the same process.

Material Orientation
Another aspect of 3D printed materials to consider is their ‘grain’ or structure in a finished part. Most 3D printed materials are not homogeneous when they are complete, i.e. they cannot support the same load in every direction because they are created in layers. This factor varies quite a bit depending the process being used and the orientation of printed parts. With FDM printing the bonding between layers is significantly weaker than the transverse directions. In sintering processes or where parts are treated in a post process after printing, the finished materials are much more homogeneous.

When a material has different Young’s modulus value depending on the direction force is applied, the material is considered to be orthotropic or transverse isotropic. When a material has a uniform tensile modulus in all directions regardless of grain then the material is considered to
be isotropic. The issue here is that no 3D printed material is fully isotropic. Some come close but since all 3D printing is done in layers there is no truly isotropic material.

Where 3D print materials are nearly isotropic then we can simulate in the shape generator using the isotropic values provided by suppliers. Where printed materials are not isotropic, as in PLA or ABS in an FDM processes, then the shape generator will lead to incorrect results that may fail under load during use. Inventor can only simulate isotropic materials in the shape generator. Regardless of your outcome every design should be validated using an FEA application. Nastran and Simulation Mechanical can both simulate non-isotropic materials.

![Figure 4 Isotropic, Orthotropic and Transverse Isotropic Materials](image)

**Figure 4 Isotropic, Orthotropic and Transverse Isotropic Materials**

**Using the Shape Generator**

While the shape generator is an automated design tool it will only produce results that are as good as the inputs provided. There is still substantial input required from the drafter/design engineer in Inventor before a valid design can be created by the shape generator.

The future of generative design promises to create entirely new geometry from your design inputs. With the shape generator in its current form, Inventor removes unneeded material based on your engineering inputs. The operator designs the volume, or the outer shape, and Inventor will remove the unnecessary mass, optimizing the existing design. The outer faces of a model can be maintained or allowed to have material removed.

Whether your part is a thin walled sheet metal part or a solid volume shape generator can create a result. The design engineer still needs to input the environmental forces, part materials and also an un-optimized solid model.
For this class we will take a closer look at a racing pedal assembly from Tilton. They supply simplified CAD models of all their products on their website. The shipped product is lightened and has material removed but the CAD models are, for the most part, un-optimized designs which makes them great for use as an example.

**Figure 5 Tilton 600-Series Floor Mount Pedal Assembly**

**Material**
If you have used Inventor Professional's stress analysis environment then, once you have loaded the Shape Generator, the Material, Constraints and Loads panels will look familiar. These are exactly the same tools used to setup a stress analysis study. Using the Assign command, you have the choice of using the material applied to the model or to override it and set your own.

**Figure 6 Assign Materials Dialog Box**
Overriding the material is a quick way that you can perform an analysis on imported geometry where a material may not have been assigned. However, a greater benefit is that it allows you, the design engineer, to experiment with alternate materials to see what result the Shape Generator will produce.

**Constraints**

There are only three types of constraints that can be applied to a model inside the Shape Generator; Fixed, Pin and Frictionless. In order to perform a shape simulation, you will need at least one constraint applied to the model at a minimum and perhaps more depending on the design you are simulating. If there are not enough constraints applied to a model an error message will appear when you attempt to generate the shape.

**Fixed**

A fixed constraint can be applied to a vertex, an edge or a face and will lock that geometry in place and also prevent it from deforming during simulation. If we were to run a simulation on the pedal base from the Tilton example you might select the surfaces of the bosses by the mounting screws as fixed surfaces.

*Figure 7 Pedal Base Constraint Locations*
Pin
A pin constraint can only be applied to cylindrical faces and prevents them from deforming in radial, axial or tangent directions. Pin constraints should be applied where these cylindrical faces are connected to adjacent geometry and allowed to rotate.

Looking back at the pedal example above, if we planned on running a simulation on the brake pedal, a pin constraint should be added to both the lower pivot point and to the connection to the brake cylinder.

Frictionless
A frictionless constraint can be applied to planar or cylindrical surfaces and prevents movement or deformation of the selected face in a direction perpendicular to the selection.

Given the forces normally applied to the pedal assembly there is no excellent example of where you might apply a frictionless constraint. However, if any lateral forces are to be applied to the pedal study then a frictionless constraint could be added to a split surface where the pedal contacts the adjacent bushing in the assembly.
Loads
Loads are the forces that are applied to the model that cause stress. Once the part model is
constrained, its ability to move or deform has been limited. With sufficient constraints you can
begin to apply forces that will act on the part.

There are five loads that can be applied to models: Force, Pressure, Bearing, Moment and
Gravity. At least one load needs to be applied to run a simulation but more can be added as
necessary.

**Force**
A standard force can be applied to individual faces or sets of faces, edges or vertices. The force’s default direction will depend on the selection but can be set in the Force
dialog box.

The force directions selected are normal to a face or along the direction of a linear edge. The direction applied can be manually overridden if no direct selection is available by
expanding the options and selecting the X, Y and/or Z vector components of the force’s
application.

Be careful with the selection sets chosen to apply forces. The magnitude of the force will
be spread evenly across the selection whether it’s a face, edge or point. The point
selection will create a highly concentrated point load which can negatively affect the
simulation results.
Pressure
Pressure is a load that can only be applied to a face selection. An equal magnitude of force is applied to the whole the face selection set in a normal direction to the face at each point along it.

Bearing
A bearing load is an application of force to cylindrical surfaces where the force is applied radially or axially. When applied in a radial direction, as shown below, force is applied to half the surface with a parabolic shaped force distribution. This keeps force from being applied to the half of the surface that isn’t being acted on when a load is applied inside the cylinder.

When a bearing direction is applied axially the force is applied to the whole surface.
Moment
A moment load is an application of force to rotate an object about an axis. The load is applied to faces only; they can be planar, cylindrical or curved selections. The direction or axis of the moment load is specified by selecting faces, edges or points.

Gravity
Gravitational force can be setup in a study when its impact is of significance. When applied gravity acts on the entire model in a selected direction. While the gravitational force value can be adjusted its default value is set to earth’s gravitational constant multiplied by the mass of your part.

Remote Force & Body Loads
In the expanded Loads panel you will find the two remaining forces that can be applied to a model; a remote force and a body load. Remote forces are applied to selected planar, curved or cylindrical faces, etc. Once a face is selected the remote point can be set. The remote point specified can be within or external to the model geometry and force is translated to the selection as a direction and moment.

A body load is an acceleration load applied either in a linear or angular direction to the entire body in the simulation. There can only be one body load applied per analysis in the Shape Generator since, like gravity, it is applied to the entire part. Selections must be surfaces or edges and will determine the direction of acceleration or the axis of velocity and acceleration the part will experience.
Goals & Criteria
Until this point in the process everything is nearly the exact same as in the Inventor stress analysis environment. The Goals & Criteria setup is the first new section that is dedicated to the shape generator. Here is where you tell Inventor what part of the model to manipulate and what the goal of the design is.

There is never going to be a case where you don’t set any goal or criteria information. So running a simulation before setting these, or at least verifying them, should be done cautiously. The first criteria you want to configure is what model geometry on the current design you want preserved.

Preserve Region
The preserve region tool works much like Inventor’s simplification tools in the sense that you have a choice between rectangular volumes and cylindrical volumes. You create these volumes to envelop geometry you want to remain unchanged. Common regions you would include in a preserve region envelope would be mass around holes, surfaces that need to be kept for other functional or aesthetic reasons. Perhaps you are generating a shape on a weldment derived to a single part; in that case you might preserve a region that encompasses an entire part in the assembly.

In the Tilton racing pedal example, three preserved regions have been added. Two at its mounting points for the pedal itself and the connection the master cylinder but also the pedal pad since it has been included in the simplified design.

Symmetry Plane
In many cases the desired part design result either done manually, or done using the shape generator, would be a symmetric model. Without setting a symmetry plane in the Shape Generator your result will be asymmetric and give results that may even be more difficult to model once returned to the part modelling environment.
To add symmetry, select the Symmetry Plane command. You have a choice of using planes about any user defined UCS feature, about the Center of Mass or Center of the part’s bounding box. You can also select more than one plane as a symmetry reference.

When symmetry is applied the Shape Generator result will be mostly symmetric but not precisely. What really matters here is symmetry applied to your part in a broader sense and not necessarily to the fine detail.

Shape Generator Settings

In the final step of the goals and criteria for your shape generation you describe the desired mass optimization, minimum member size and mesh resolution.

When setting your target mass, you have a choice of a percentage reduction of mass or a specific target value. Keep in mind that both of these values include mass inside any preserved regions. If you enter a target mass below what is possible to reach the safety factor based on the other study settings Inventor will alert you when you try to generate the shape.

One of the struggles you will have with shape generator is with the minimum member size and the mesh resolution. Turning on minimum member size means that Inventor will attempt to connect every branch in the mesh result. This isn’t always possible and it may take a few simulations before a satisfactory result is achieved.
Autodesk recommends that the minimum member size is no smaller than three times the average mesh element size. Size settings underneath this threshold will produce a warning message that can be ignored if desired but your results will likely suffer.

There will often be a balance between mesh size and the minimum member size. A finer mesh does not necessarily mean a better shape but if you desire a smaller setting for the minimum member size your mesh must be at least 1/3 of that size. The finer the mesh the longer an analysis will take.

Based on the materials, the constraints, the forces and our rules the Shape Generator has enough information to create a model, or to tell you if there is no possible solution.

Run
With the study criteria set all that should be left is to create the mesh and run the Generate Shape command. To create a mesh simply click the Mesh View button. After a quick process your model display should change from the solid, parametric, view to the mesh view.

Next click Generate Shape and select the Run button to perform the analysis. At this point if your model is under constrained or the amount of material cannot be removed based on your mass target you will receive errors in the expanded run panel. Any error must be corrected before the analysis can be run. Warnings that pop up during a shape generation will not stop the analysis but should be interpreted to understand if they are a concern.

Depending on the model complexity and mesh size you can expect the shape generation to take some time. The generation process will use all cores available on your computer's CPU and times can range from minutes to hours. Once complete you should see a mesh shape result appear in the window.
Use the Mesh to Generate a Parametric Model

If the result returned by the Shape Generator is satisfactory then the next step is to export this shape for use in your design. Depending on the manufacturing process you are using and the design of the part there are a couple of options. You can export the resultant shape created directly to the part modelling environment for use as a modelling guide or you can save the result as an STL file format.

Used as a Guide

If you plan to use traditional manufacturing methods to create your part then the shape produced from the simulation should be exported to your modelling environment. From the Shape Generator tab, use the Promote Shape command and select 'Current Part File'. This pushes the mesh result into your parametric model where it can be used to help you fine tune the original design. The catch is that the Shape Generator simulation produces a mesh result and meshes cannot be manipulated or modified by Inventor.
When promoted, the mesh is intended to be used as a rough modelling guide. Using standard sketching and feature tools you then draw cutouts based on the area shown as being removed by the promoted mesh which is overlaid onto your part.

This is an ideal process for sheet metal or plate parts manufactured using CNC punching, laser, waterjet processes etc. You can be as rough or accurate as you like using this method just be sure to validate the finished design using Inventor's stress analysis environment or similar.
Parts that don’t fall into the sheet metal or plate category can still use this manual process of removing material however it will be significantly more work. While more work and an increase in the level of difficulty you can use more than just standard sketching and feature tools. Consider using the freeform tool to push/pull geometry that will get removed from the inside of your part. Then the Combine command can be used to remove that volume from the model. Another option, as we will see below, is to use surface geometry.

**FIGURE 22 VOLUMINOUS PART WITH MESH AS A MODELLING GUIDE**

**Sketch with Mesh Projections**

With Inventor 2017 there are some additional tools available for sketching using mesh geometry that were not available with the 2016 R2 release. If you would like to manually remove material using the mesh as a guide but would prefer to be more accurate, based on the Shape Generator’s result, then you can project mesh elements directly into a sketch for use. In this case you can select mesh vertices, edges or faces directly in the sketch used to produce cutouts or construction lines.

**FIGURE 23 SKETCH PROJECTIONS FROM MESH ELEMENTS**
You can see the selectable mesh objects in the Select Other dialog box shown. Planar mesh faces, mesh edges and mesh vertices can all be selected as sketch projections.

The only difficulty with this process is that projected mesh edges and faces will often be self-intersecting. This means that selecting mesh edges in order from left to right, as shown above, is often not possible. The best case would be to select every second element or random elements to project as construction lines. These construction lines can then be used as a guide. A better potential option is to project the mesh vertices and simply draw lines to ‘connect the dots’.

**FIGURE 24 CUTOUT WITH MESH PROJECTIONS**

**Surfacing with Mesh Geometry**
Another option that you have with Inventor 2017 is to use the new Fit Mesh Face command. This tool allows you to select mesh facets and derive surface geometry from them. The surfaces created with this tool can then be used with standard modelling operations. You might use surfaces created from a mesh to trim, offset or stitch to create or modify a parametric part model.

Starting the Fit Mesh Face command, you select the size of your brush and the accuracy of the new surface. The brush size selected works as a tool to select and highlight mesh facets. Its size is set via the slider but is always relative to your screen size and not the model. Once you start painting (selecting) facets Inventor will attempt to create a surface from the selection. An automatic method is possible or the user can explicitly select from a Plane, Cone, Sphere, Torus or Spline surface generation type.
The sketching tools and the Fit Mesh Face command are all discussed here for use on meshes created by the Shape Generator; However, they can certainly be used with other design workflows. Any mesh, created by any means, that has been imported into Inventor’s part modelling environment can make use of these. Another excellent example might be the use of scanned data, or geometry created/imported from non-parametric modelers such as animation software.

**Printing the Mesh Result**

The Shape Generator’s capabilities to this point are exciting. However, in the section above there is quite a roadblock to manually creating a part model by removing geometry if you are creating anything but a flat part.

Two aspects of the workflow conspire against us with voluminous designs. First, as identified is the modelling approach which is difficult and not optimal for creating organically shaped volumes to remove from your design. The second is your manufacturing process; Even if you were able to remove geometry from your part exactly as the Shape Generator displayed you would not likely be able to manufacture that part using traditional means.

This leads us to the really exciting part of generative design technology and the reality it presents with 3D printing technologies. IF (and it’s a big ‘if’ at this time) you get a suitable mesh result from the shape generator you can use the ‘Promote Shape To: STL File’ to produce 3D printed parts without any further modelling effort.

**Print the STL Mesh**

There is not a lot new here once you have an STL file that is suitable for consumption in a 3D printing process. What you do have to watch for, as mentioned earlier, is that the result
achieved by the Shape Generator makes sense. Generative design is far from perfect, for now, and still often needs checks and tweaks to create a valid design. You will need to verify and/or manipulate your constraints, forces and goals a number of times to get a result that does not have any hanging or floating mesh sections. While you might be able to model around these problems in a manual practice; if you decide to go straight to printing the mesh file multiple attempts and final verification will need to be done before getting a suitable part. Keep the minimum member size as high as possible, relative to your mesh size, to reduce the likelihood of floating mesh objects.

**Figure 26 Mesh with Floating Elements**

Use AnyCAD to Place in an Assembly
If you have decided to produce the generated shape directly without any further modelling other challenges may arise. i.e. you likely still need to represent the part in a larger assembly and perhaps create a drawing with some detailed views. If you require the latter, then unfortunately, there is no ability in Inventor yet to create drawing views of mesh models. For the former, Inventor 2017 introduces new capabilities so that meshes can be included and constrained in your assemblies.

**Figure 27 Mesh Part Placed in the Pedal Assembly**
STL files can be placed directly in an assembly using Inventor’s AnyCAD functionality. The STL file type is one that needs to be converted so in the end an IPT will be saved with the mesh from the STL as a node in its feature tree. Like the sketch enhancements though, you have the ability to select mesh faces, edges or vertices using the Constrain or Joint tools to fully control the mesh’s position relative to your other components.
Since the mesh parts are not solid models they do not have yet a calculable mass value in Inventor. If you decide to use mesh files directly you will have to provide a mass override (and override any other physical properties) to get an accurate value for the total assembly. There is also no option shrinkwrap or derive a mesh to create a solid within Inventor. These limitations aside you have a part file, can include it in an assembly and still set its physical values. Not much more is typically necessary in design.

**Meshmixer**

There is another tool within Autodesk’s portfolio that can be helpful if you are using the STL files directly in your assemblies. Meshmixer is a free downloadable application that can be used to edit mesh files.

Using Meshmixer you can smooth mesh facets and perform a host of other operations on mesh files. The goal with this tool would be to make minimal changes to a mesh. If any large scale changes are needed, then the time savings and analysis that the Shape Generator has done are devalued. Once a mesh is finished it can be re-exported to an OBJ or STL file that can be taken back into Inventor and modelled using the same tools shown above.

Finally, another great reason to use Meshmixer is to use it to send your files to a 3D printer. Meshmixer can use a number of printers that are preset or link to online printing services like...
Shapeways. Once connected to Shapeways you can select a material, check the price and order directly from the application, bypassing the website, creating a seamless process.

![Figure 31 Direct Printing to Shapeways](image)

**Summary**

Generative design technologies are still young and there are challenges that we have seen in the tool’s current form with generating contiguous meshes, analyzing mass, and detailing on drawings. However, the technology’s development is rapid and there is a convergence of generative design and 3D printing that has made a whole new design process a reality.

Today, you can use generative design to optimize your part files, automatically creating their final geometry. Then, take that shape generated, place it in an assembly and manufacture the part(s) without further modelling or manipulation. Imagine what's in store for this technology as it develops and matures in the years to come.