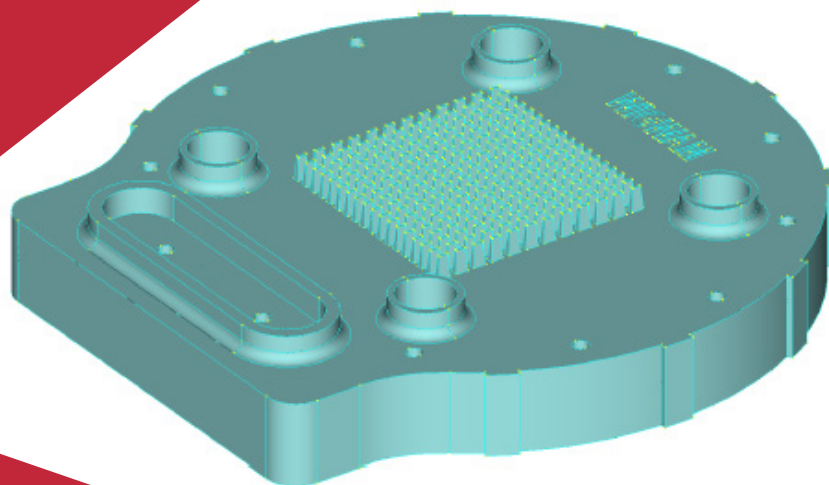


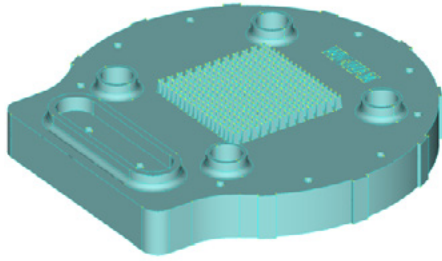
Femap II.I

WHITEPAPER

Geometry Simplification and Preparation in Femap



Geometry Simplification and Preparation in Femap



Software:
Femap 11.1

Overview

Part geometry is often complex and may require idealization before a finite element model can be built. Femap contains a suite of geometry preparation tools to simplify geometry before a mesh is applied and guide the mesh in the location of interest. This document discusses some of the powerful geometry preparation and simplification tools available in Femap.

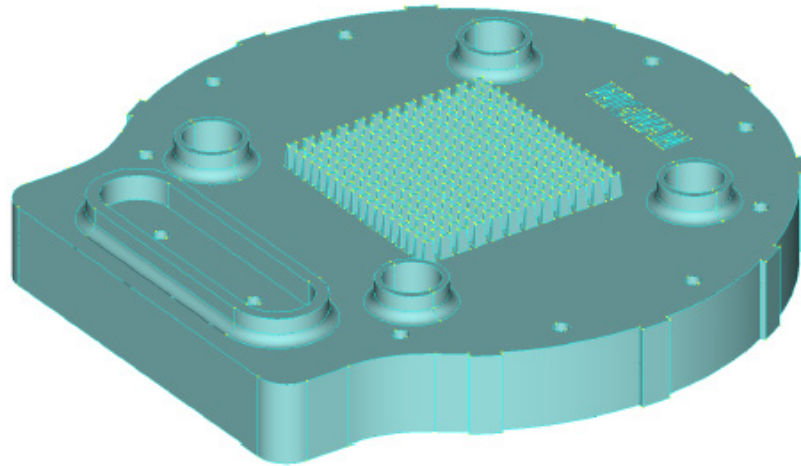
This whitepaper is part of a series of free Siemens PLM Software training resources provided by ATA. For more whitepapers, tutorials, videos, and macros, visit ATA's PLM Software website: <http://www.ata-plmsoftware.com/resources>.

Geometry Simplification and Preparation in Femap

Geometry Simplification

CAD parts are often too complex for an analyst to mesh easily; geometry simplification may be necessary. Femap contains a suite of geometry idealization tools to simplify geometry and prepare it for meshing. Most of the geometry preparation tools are found under the Geometry dropdown menu. In this example, we will be simplifying the electronics baseplate shown in Figure 1.

Figure 1: ►
Electronics baseplate geometry.



This part contains many features that are unimportant to the analysis. The imprinted text, small exterior holes, and side ribs will distort elements and complicate the mesh. These features will be removed for the FEA model.

First, we will remove the side ribs that line the circular portion of the baseplate. Using Geometry → Solid → Remove Face, select the three faces on each rib and click OK. Figure 2 shows the selected ribs and the resulting baseplate. The ribs have been removed and the cylindrical portion of the baseplate remains. The rib edges were also removed.

Geometry Simplification and Preparation in Femap

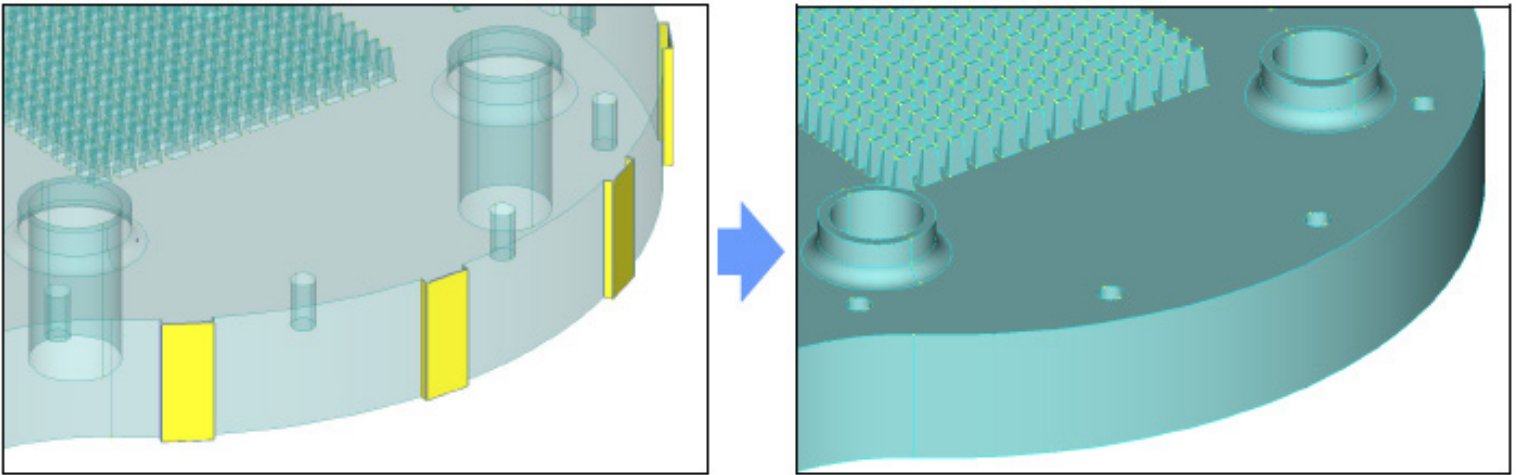


Figure 2: ▲
Removing side ribs with Remove Face.

We will also use the Remove Face command to remove the faces of the embedded text. While pressing Shift, drag the mouse over the faces that make up the text. Deselect the faces not defining the text and click OK (Figure 3). In general, the Remove Face command works best when removing a small number of faces at a time. Sequential Remove Face executions with fewer faces selected may produce a more desirable final solid.

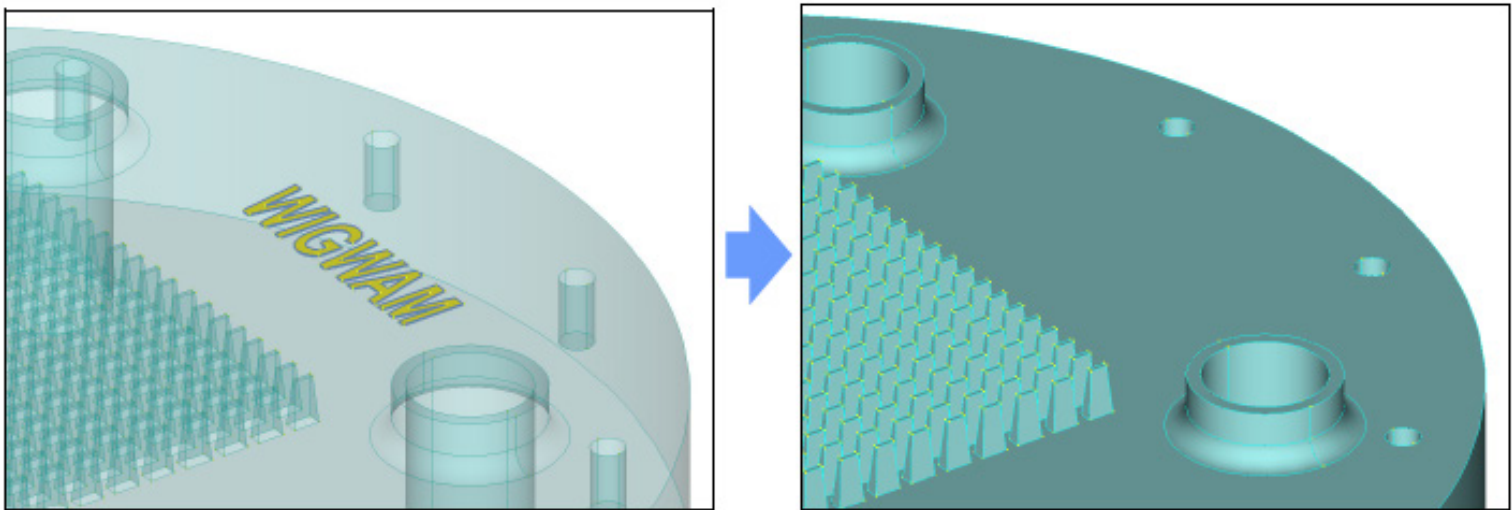
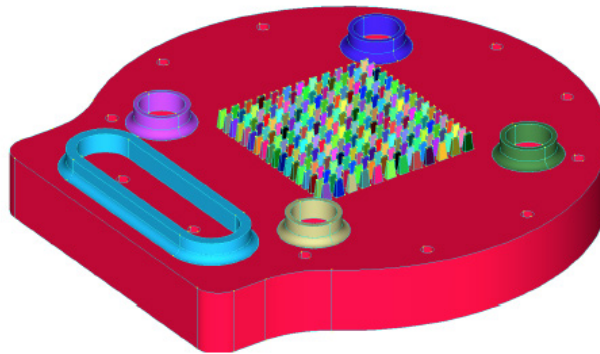


Figure 3: ▲
Removing indents with Remove Face.

Geometry Simplification and Preparation in Femap

Next, we will remove the heat fins from the baseplate, as they are also unnecessary for a structural analysis. The Slice command (Geometry → Solid → Slice) creates multiple solids by cutting a single solid along a plane. To define your cut plane, select three points on the base of a fin. You may also use another plane creation commands by clicking on the Methods button. Figure 4 shows the various solids created by the Slice Along Face command with each solid colored differently: each fin is now a separate solid. The Slice Along Face command uses an existing geometric face to slice along. We selected the top face of the baseplate

Figure 4: ►
Multiple solids after the Slice command.



Once the baseplate and the fins are separate solids, the fin solids can be deleted using Delete → Geometry → Solid. Six separate solids still remain, as the Slice command also split off the mounting hole ribs. Although all of these solids originated from a single solid, they will mesh separately and be unconnected in the FEA model unless further action is taken: add the solids together with Geometry → Solid → Add so they will be meshed as one.

This can also be done while meshing individual solids by using the Adjacent Surface Matching option in Mesh → Mesh Control → Size On Solid. This option forces adjacent surfaces on different solids to share nodes. If you use this option, ensure that the surfaces touching are the same shape and have the same prescribed element size.

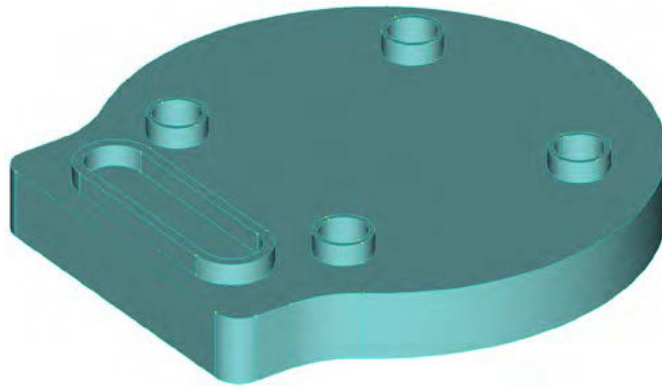
Next, we will remove the outer ring of small tapped holes. Since they are not through-holes, the top and bottom faces of the baseplate are not identical. This prevents a 2-D mesh on the top face from being extruded through the thickness to create 3-D brick elements. The holes can be removed using Geometry → Surface → Remove Hole. Select a curve on each hole and click OK.

Finally, the fillets on the mounting ribs will complicate the mesh. We can remove the fillets using a tool in the Meshing Toolbox, which is accessed by clicking on the Meshing button on the Panes toolbar. The Meshing Toolbox contains many geometry preparation and meshing tools that simplify the model creation process.

Geometry Simplification and Preparation in Femap

Some of the tools in the Meshing Toolbox have analogous tools in the Geometry dropdown menu. In this case, we will be using Feature Removal → Surfaces command in the Meshing Toolbox, which is analogous to the Remove Face command we used previously. Click on the Dialog Select button at the top of the Meshing Toolbox, select all of the fillet surfaces, and click OK. The baseplate should now look like Figure 5.

Figure 5: ►
Solid body after geometry idealization.



Preparing Geometry for a Mesh

Now that our solid is simplified, we want to split the surfaces on the top of the baseplate. This will help guide the mesh in regions of importance. For example, we removed the fillets on the hole ribs. In the resulting sharp corners, we want consistent elements that will provide accurate stress or strain estimations for a weld or corner concentration factor.

To begin, we will be using the Geometry Editing → Washer command in the Meshing Toolbox. This command creates a concentric circle at a different diameter than an existing curve, which helps the mesher create a consistent ring of elements around the ribs. Enter an offset distance, click the Dialog Select button at the top, and click the curves you wish to copy. You can also use the Select button, which will create the features on the fly. In this case, we will create two concentric curves around each mounting hole rib.

The previous step creates nodes in a concentric ring around the holes but does not guarantee that the mesher will create elements normal to the mounting holes. Splitting these rings into four quarter circles helps the mesher make elements square to the curves. Using Geometry Editing → Curve Break, split the half circles into quarters by entering a point very far away in the desired direction. Femap will project the point onto your curve and split at that location.

Geometry Simplification and Preparation in Femap

After the curves are split into quarters, use Geometry Editing → Point to Point to create a curve connecting the new points. You may have to regenerate (Ctrl + G) your model to see the newly created points. This will force element edges to align normal to the curves. The baseplate should now look like Figure 6.

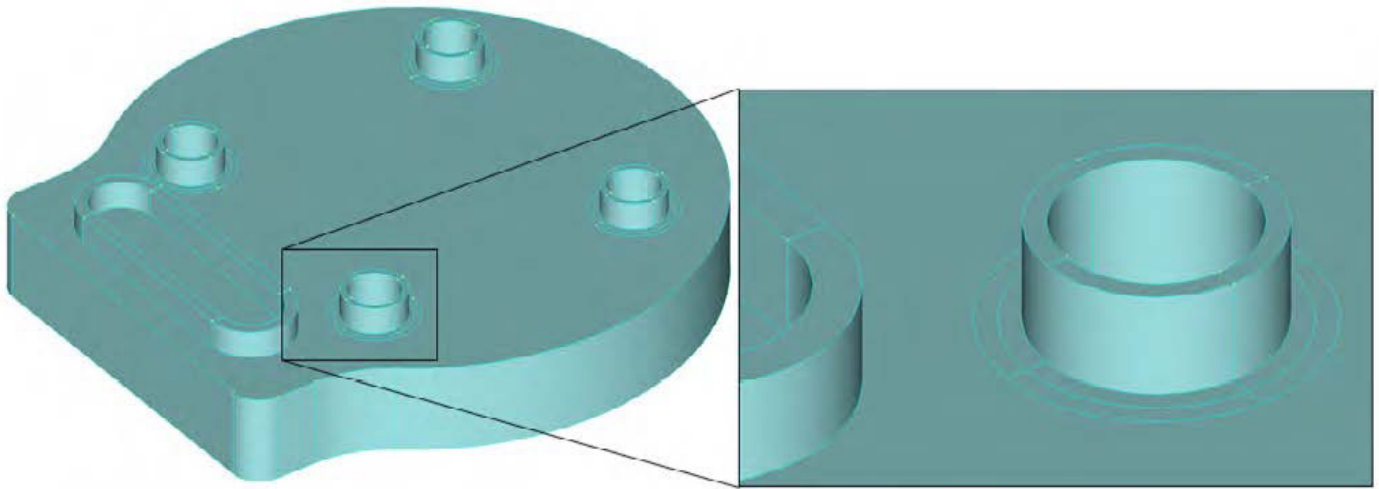


Figure 6: ▲
Solid body after edge splitting

If desired, you can also combine curves or surfaces from within the Meshing Toolbox. Under Combined / Composite Curves and Combined / Composite Surfaces, you can remove points or curves to create single curves or surfaces, respectively. When curves and points are removed, the mesher will not be forced to build element edges or nodes at those locations.

More information on Geometry Preparation can be found in the Femap Help under Mesh, Geometry Preparation. More information on the Meshing Toolbox can be found in the Femap Help under Tools, Meshing Toolbox.

www.ata-plmsoftware.com

ATA Engineering



Solution
Partner

PLM

SIEMENS

-  www.ata-e.com
-  [ata-engineering](https://www.linkedin.com/company/ata-engineering)
-  [@ataengineering](https://twitter.com/ataengineering)
-  sales@ata-e.com
-  858.480.2000

Copyright © ATA Engineering, Inc. 2018

www.ata-plmsoftware.com

San Diego
Corporate Headquarters

Albuquerque

Denver

Huntsville

Los Angeles

San Francisco

Washington, D.C.