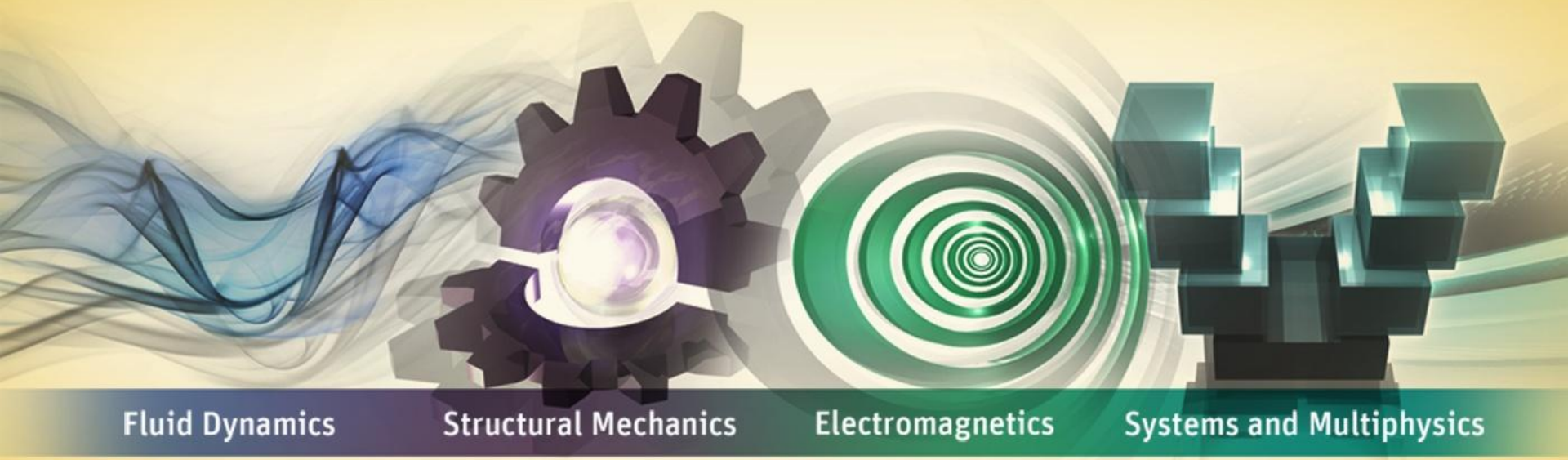
ANSYS

Realize Your Product Promise™

Workshop 8:
Basic Meshing Operations



ANSYS Maxwell 3D V16

About Workshop

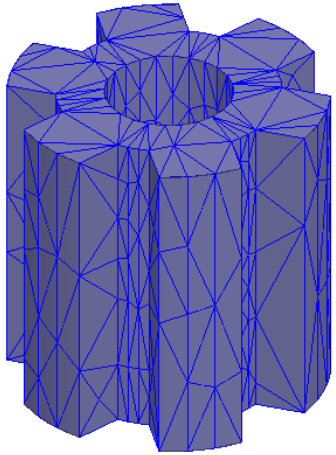
- **Introduction on Meshing Operations**
  - Meshing in Maxwell is highly automated. Adaptive meshing algorithm employed in Static solvers of Maxwell does automatic refinement of mesh depending upon solution convergence. Thus in most of the static cases there is no necessity of applying mesh operations.
  - Transient Solvers on the other hand, does not use adaptive meshing technique. Although Maxwell offers capability of transferring adaptively refined mesh from Static Solver to Transient solver, this may not be possible for all scenarios. Thus we have rely on mesh operations. Even in static solvers as well some times mesh operations are assigned to reduce number of adaptive passes carried out to achieve required accuracy.
  - This workshop introduces the different mesh operations that Maxwell offers. Mesh operation are used either to cut the number of adaptive passes or to optimize the mesh details on complicated objects. We will illustrate the meshing operations using an SRM core.

Problem Setup

- **Create Design**
  - Select the menu item ***Project → Insert Maxwell 3D Design***, or click on the  icon
- **Create Core**
  - Select the menu item ***Draw → User Define Primitive → RMxprt → SRMcore***
  - In User Defined Primitive Operation window,
    1. Press **OK** to accept default settings
  - Change the name of the Object to **Core**
- **Save File**
  - Select the menu item ***File → Save As***
    1. Save the file with name **WS8_BasicMeshingOperations.mxwl**
- **Create Analysis Setup**
  - Select the menu item ***Maxwell 3D → Analysis Setup → Add Solution Setup***
  - In Solve Setup window,
    1. Press **OK** to accept default settings

Generate Initial Mesh

- **Create Initial Mesh**
  - Select the menu item ***Maxwell 3D → Analysis Setup → Apply Mesh Operations***
- **Plot Mesh on the Object**
  - Select the Object **Core** from the history tree
  - Select the menu item ***Maxwell 3D → Fields → Plot Mesh***
  - In Create Mesh plot window,
    1. Press **Done**



- **View Statistics of Mesh**
  - Select the menu item ***Maxwell 3D → Results → Solution Data***
  - In Solutions window,
    1. Select the tab **Mesh** Statistics

Profile | Convergence | Force | Torque | Matrix | Mesh Statistics

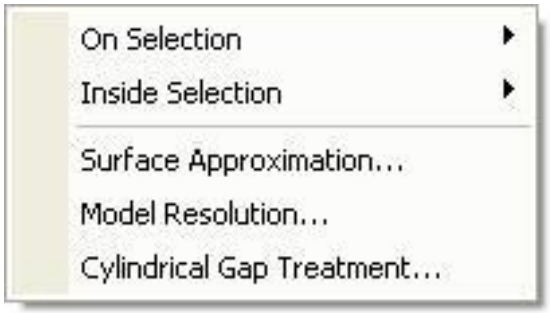
Total number of mesh elements: 1558

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol
Core	1558	10.6137	68.2854	23.7959	20.4456	2356.2

Note : This is initial mesh created by Maxwell. Apart from transient solver, for all other solvers this mesh will be refined for each pass in order to increase accuracy of solution. For Transient solutions, mesh refinement is not done. So it is required to apply appropriate mesh sizes which will be discussed in next few slides.

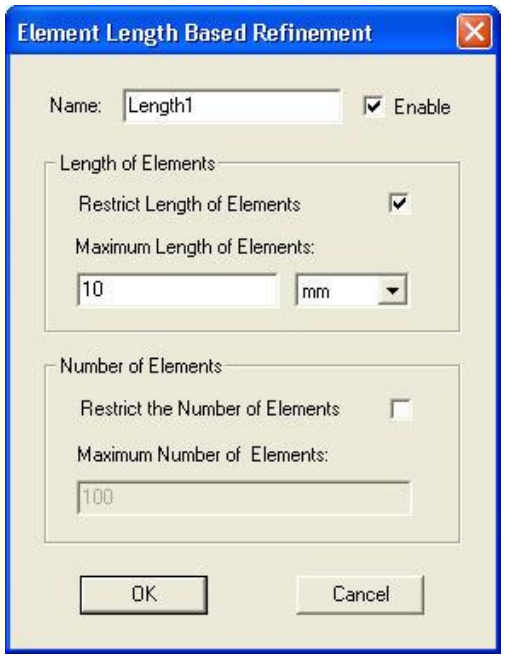
Mesh Operations

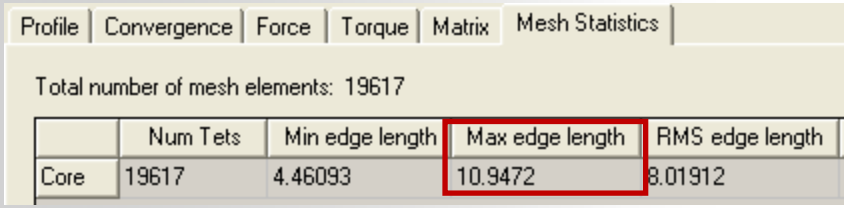
- **Mesh Operations**
  - Maxwell3D has number of meshing operations to construct a better initial mesh for pass one. These mesh operations enable you to add elements in a volume, on a surface. It is also possible to monitor the discretization of true surfaces.
  - Note that when assigning a mesh operation, the mesh process will do its best to cope with the specified operations. Automatic meshing is a complex operation, so other constraints are to be taken into account. As a result, sometimes, the obtained mesh is not quite exactly what has been entered because internal constraints (seeding, facetting, ..) have imposed a different mesh topology.
  - For selected objects, five different type of mesh operations can be applied. First two operations are used to add elements either on the surface or in the volume of the objects. The last three meshing operations will monitor the way the meshing will handle the geometry of the object.



Mesh operation: Inside Selection

- **Apply Mesh operations inside an Object**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D → Mesh Operations → Assign → Inside Selection → Length Based***
- **Restrict Length of Elements**
  1. Restrict length of Elements: ☑ **Checked**
  2. Maximum Length of Elements: **10 mm**
  3. Restrict the Number of Elements: ☐ **Unchecked**
  4. Press **OK**
  - Generate Mesh with above Settings

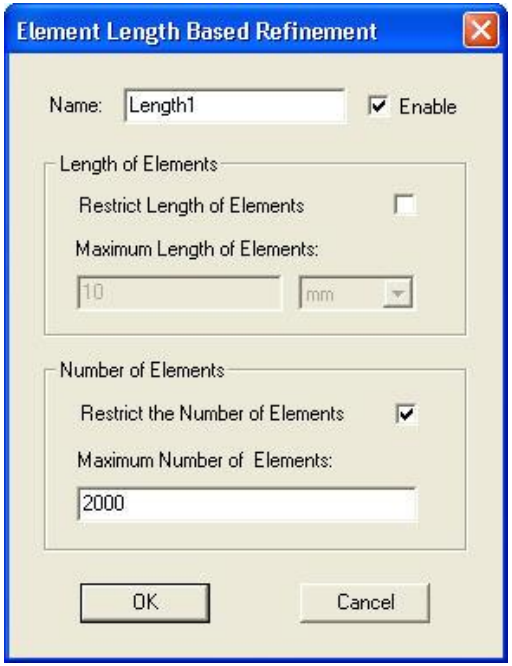


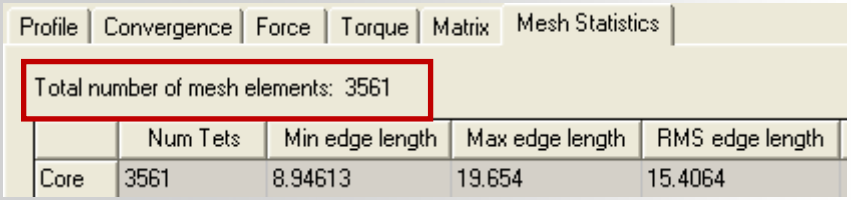


Note: Inside Selection Mesh operation indicates that mesh refinement will be carried out inside the volume of the selected object

Mesh operation: Inside Selection(*Contd..*)

- **Restrict Number of Elements**
  - Select the menu item ***Maxwell 3D → Analysis Setup → Revert to Initial Mesh***
  - Expand the Project Manager tree to view **Mesh Operations**
  - Double click on **Length1** to modify its parameters
  - In Element Length Based Refinement window,
    1. Restrict length of Elements: ☐ **Unchecked**
    2. Restrict the Number of Elements: ☑ **Checked**
    3. Maximum Number of Elements: **2000**
    4. Press **OK**
  - Generate Mesh with above Settings

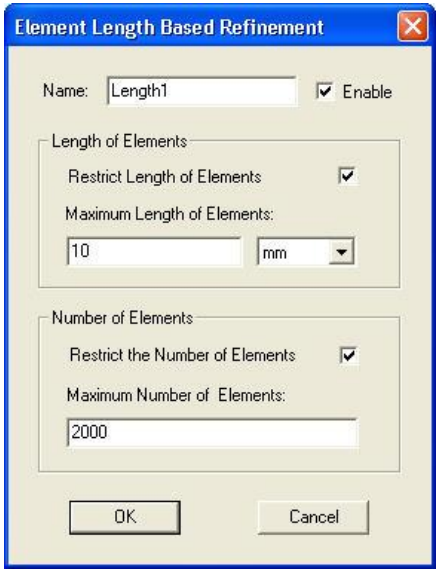


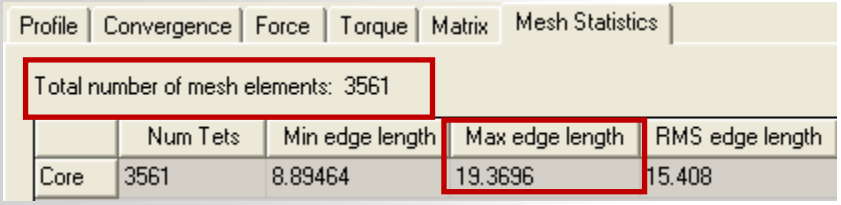


Note: Initial mesh had 1558 elements. This shows that mesh operation has added around 2000 elements to the object.

Mesh operation: Inside Selection(*Contd..*)

- **Restrict Length and Number of Elements**
  - Select the menu item ***Maxwell 3D → Analysis Setup → Revert to Initial Mesh***
  - Expand the Project Manager tree to view **Mesh Operations**
  - Double click on **Length1** to modify its parameters
  - In Element Length Based Refinement window,
    1. Restrict length of Elements: ☑ **Checked**
    2. Maximum Length of Elements: **0.55 mm**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **2000**
    5. Press **OK**
  - Generate Mesh with above Settings

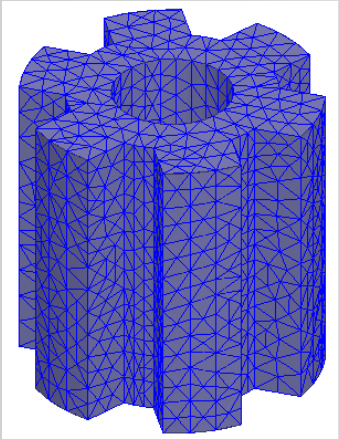




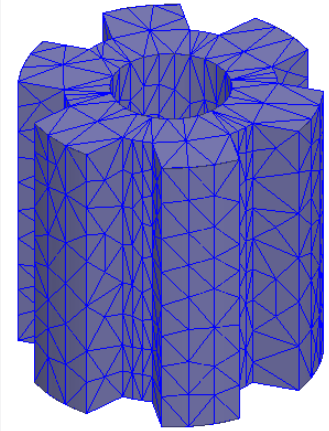
Note: First condition was ignored as the mesh refinement reached second condition

Mesh operation: Inside Selection(*Contd..*)

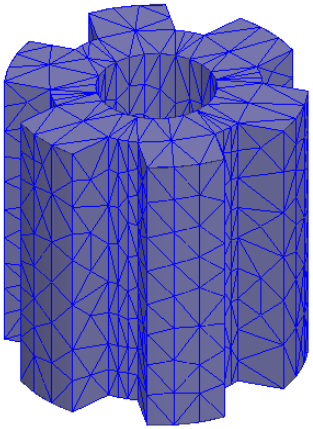
- **Comparison**
  - When used Restrict Length of Elements option, maximum length of the elements created in the object will be restricted to the specified value. Mesh will be refined until the element sizes in the object reach specified value or lower.
  - When Restrict Number of Elements, mesh will be refined until added number of elements in the object reaches the specified threshold
  - When both number of elements and length of elements is restricted, Maxwell will stop when the first constraint is attained. In this particular case, entering only a maximum length of 10mm produces about 20,000 elements. Therefore, having both 10mm and 2,000 elements lead to have only 2,000 elements added.



Restrict Length of Elements



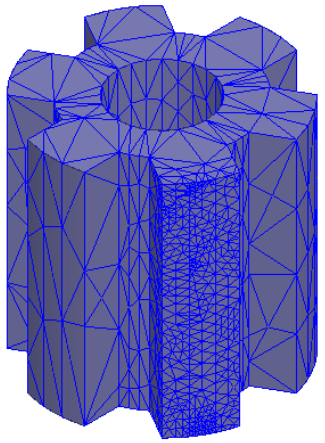
Restrict Number of Elements



Both

Mesh operation: On Selection

- **Apply Mesh Operations on a Surface**
  - Delete already applied mesh operations
  - Select the menu item ***Maxwell 3D → Analysis Setup → Revert to Initial Mesh***
  - Select the menu item ***Edit → Select → Faces***
  - Select any face of the object
  - Select the menu item ***Maxwell 3D → Mesh Operations → Assign → On Selection → Length Based***
  - In Element Length Based Refinement window,
    1. Restrict length of Elements: ☐ **Unchecked**
    2. Restrict the Number of Elements: ☑ **Checked**
    3. Maximum Number of Elements: **2000**
    4. Press **OK**
  - Generate Mesh with above Settings



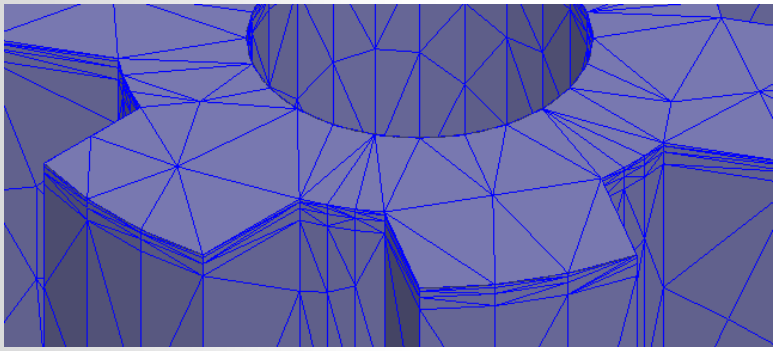
Note: On selection Mesh Operation will refine the mesh on the selected surface. If object is selected for above mesh operation, refinement will be done on the surfaces of the object. The refinement control can be achieved by either restricting length or number of elements as described earlier

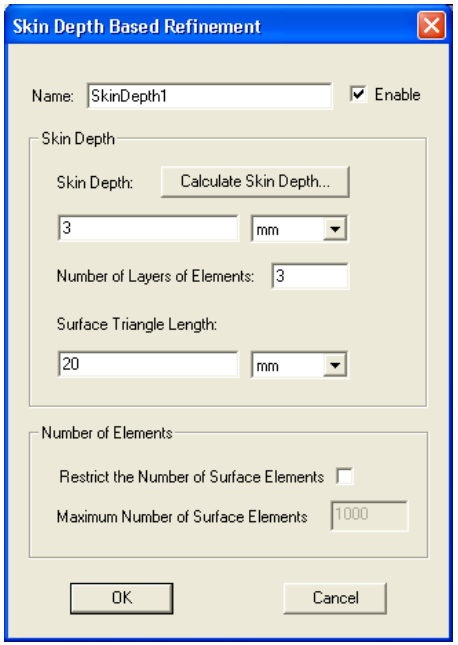
Mesh operation: On Selection(*Contd..*)

- **Skin Depth Mesh**
  - This mesh operation has to be used with great caution. The goal is to impose to Maxwell to mesh in the skin depth with a given number of layers and a given mesh density.
  - The major consequence of the mesh operation is that the adaptive meshing process does not have the full freedom when refining the mesh. Usually, it is preferred to use phantom objects or shells to take into account skin depth.
  - The skin depth mesh can be useful in capturing eddy currents in the geometry where skin depth is very small compared to thickness of the object. In such cases, there is a risk that adaptive meshing may not able to capture the skin depth correctly or requires lot of elements and adaptive passes to provide correct results.

Mesh operation: On Selection(*Contd..*)

- **Assign Skin Depth Mesh Operation**
  - Delete already applied mesh operations
  - Select the menu item ***Maxwell 3D → Analysis Setup → Revert to Initial Mesh***
  - Select the top face of the core
  - Select the menu item ***Maxwell 3d → Mesh Operations → Assign → On Selection → Skin Depth Based***
  - In Skin Depth Based Refinement window,
    1. Skin Depth: **3mm**
    2. Number of Layers of Elements: **3mm**
    3. Surface Triangle Length: **20mm**
    4. Restrict Number of Elements: ☐ **Unchecked**
    5. Press **OK**
  - Generate Mesh





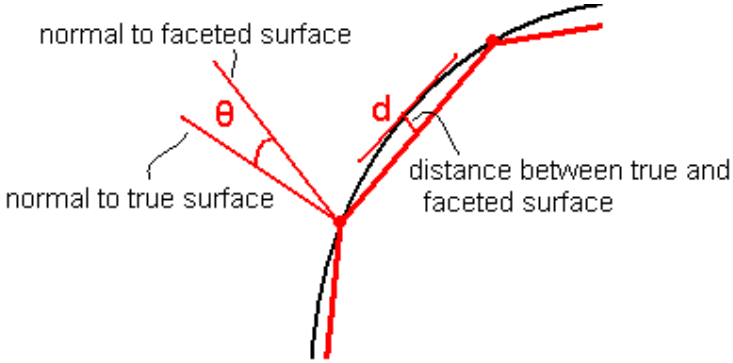
Mesh operation: Surface Approximation

- **Surface Approximation**
  - This mesh operations control the way Maxwell handle true surfaces. When Curved surfaces are meshed, an approximation of geometry is done as mesh can not capture all points of the curved surface. The deviation of mesh from geometry can be measured using two parameters

Surface Deviation: On Curved surfaces, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units.

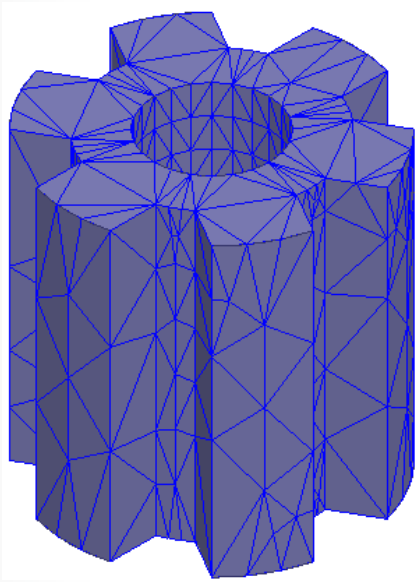
Normal Deviation: The angular difference between the normal of the curved surface and the corresponding mesh surface at the same location is called the *normal deviation* and is measured in degrees (15deg is the default).

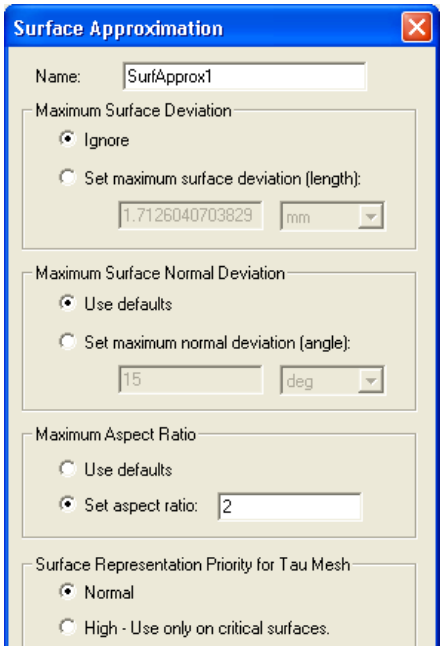
  - Users can control mesh approximation be defining these two parameters



Mesh operation: Surface Approximation (*Contd…*)

- **Apply Surface Approximation**
  - Delete applied mesh operations and Select the Object **Core** from the history tree
  - Select the menu item ***Maxwell 3D → Mesh Operations → Assign → Surface Approximation***
- **Set Maximum Aspect Ratio**
  - In Surface Approximation window,
    1. Set aspect Ratio : **2**
    2. Press **OK**
  - Generate Mesh



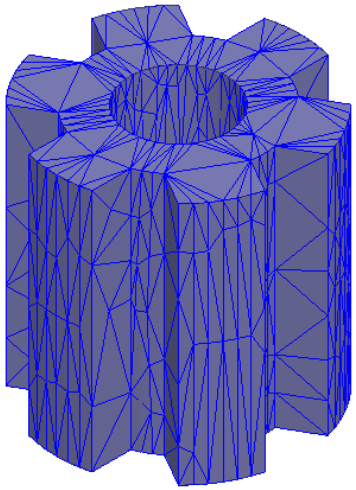


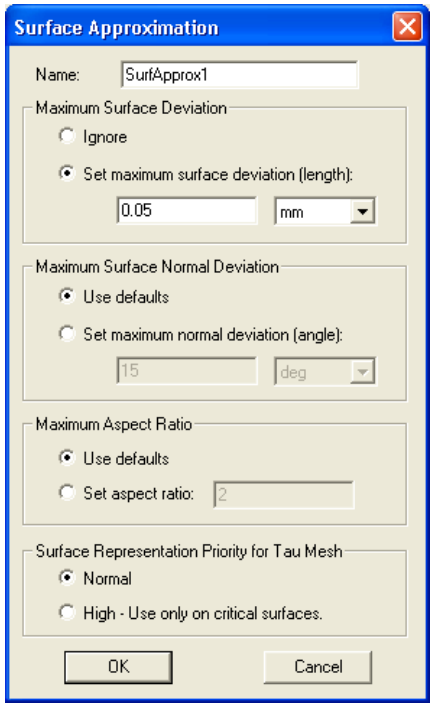
Note: Changing the Maximum Aspect Ratio will not affect curved surface mesh but will control aspect ratio of generated elements and ensure the mesh quality.

Setting Priority for Tau Mesher will set tolerance in capturing small features. High option will create better mesh on small details with expense of mesh count.

Mesh operation: Surface Approximation (*Contd…*)

- **Set Maximum Surface Deviation**
  - Expand the Project Manager tree to view **Mesh Operations**
  - Double click on **SurfApprox1** to modify its parameters
  - In Surface Approximation window,
    1. Maximum Aspect Ration: set to **Use Default**
    2. Set maximum surface deviation (Length): **0.05mm**
    3. Press **OK**



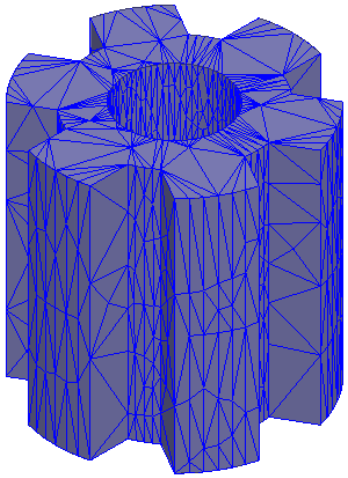


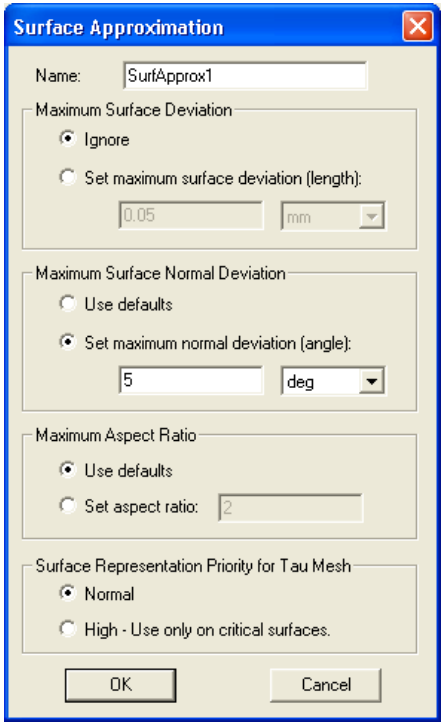
Note: Setting maximum Surface Deviation will control the curved surface approximation by controlling parameter d shown in image on slide 13.

Unlike other mesh parameters which refine already created mesh, Surface approximation parameters are applied while creating initial mesh itself.

Mesh operation: Surface Approximation (*Contd…*)

- **Set Maximum Surface Normal Deviation**
  - Expand the Project Manager tree to view **Mesh Operations**
  - Double click on **SurfApprox1** to modify its parameters
  - In Surface Approximation window,
    1. Maximum Surface Normal Deviation: set to **Ignore**
    2. Set maximum normal deviation (angle): **5deg**
    3. Press **OK**



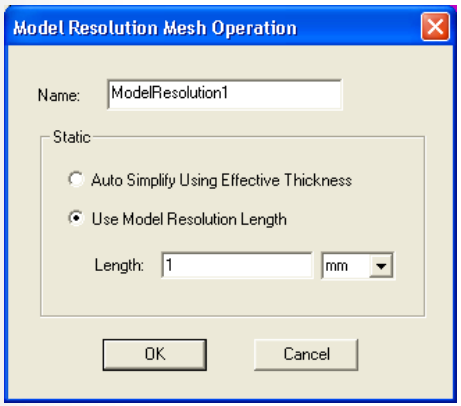


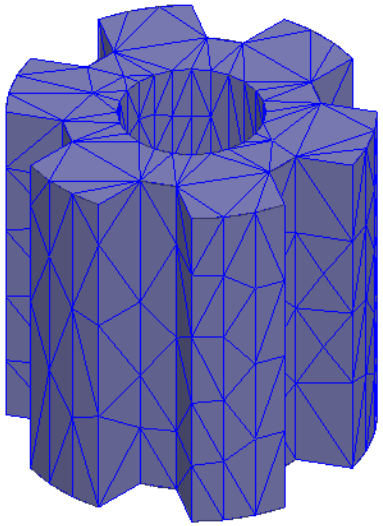
Note: Setting maximum Surface Deviation will control the curved surface approximation by controlling parameter θ shown in image on slide 13.

Setting 5deg angle will generate 3 times more elements on curved surface than default value which is 15deg

Mesh Operation: Model Resolution

- **Set Model Resolution**
  - Delete the previous mesh operation
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D → Mesh Operations → Assign → Model Resolution***
  - In Model Resolution mesh Operation window,
    1. Set the tab to **Use Model Resolution Length**
    2. Set the Length value to **1mm**
    3. Press **OK**





Note: This operation is meant to disfeature an object. By putting 1mm, Maxwell will disregard any detail below 1mm. This can be useful when dealing with imported geometries where some details, inconstancies remain. This operation is seldom used. For selected geometry, the result is not spectacular at all.

Mesh Operation: Cylindrical Gap Treatment

- **Cylindrical Gap Treatment**
  - These mesh operations are generally used to resolve narrow gaps in motor geometries between stator and rotor. The Band region created between them needs to have fine mesh resolution. In such cases it is helpful to use this mesh operation on Band region.
  - This mesh operation is created by default on Band object when motion is assigned
- **Create Band Region**
  - Delete existing mesh operations
  - Select the menu item ***Draw → Cylinder***
    1. Using the coordinate entry fields, enter the center of the base
       - **X: 0, Y: 0, Z: -60**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius and height
       - **dX: 51, dY: 0, dZ:120**, Press the **Enter** key
  - Change the name of the object to **Band**

Mesh Operation: Cylindrical Gap Treatment (*Contd…*)

- **Assign Cylindrical Gap Treatment**
  - Select the object Band from the history tree
  - Select the menu item ***Maxwell 3D → Mesh Operations → Assign → Cylindrical Gap Treatment***
  - Generate Mesh
  - The refinement will be done in the region of small gap in order to capture the thin region.