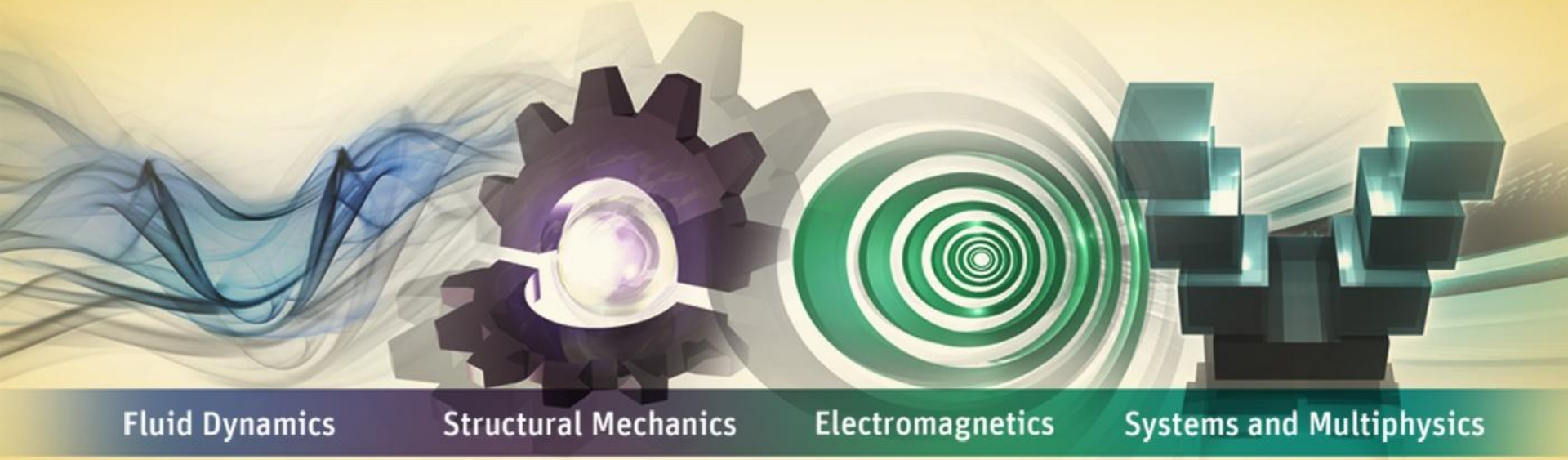
ANSYS

Realize Your Product Promise™

Lecture 6:
Meshing and Mesh Operations



ANSYS Maxwell V16
Training Manual

Content

A. Maxwell Meshing

B. Initial Meshing

C. Adaptive Meshing

D. Mesh Operations
   a. On Selection
   b. Inside Selection
   c. Surface Approximation
   d. Model Resolution
   e. Cylindrical Gap Treatment

E. Applying Mesh Operation

F. Mesh Linking

G. Troubleshooting

A. Maxwell Meshing

About Mesh

- Maxwell uses the Finite Element Method (FEM) to solve Maxwell's equations.
- In order to obtain the set of algebraic equations to be solved, the geometry of the problem is discretized automatically into basic building blocks(e.g., tetrahedra in 3D).
- The assembly of all tetrahedra is referred to as the finite element mesh of the model or simply the mesh.
- Mesh plays important role in accuracy of the computed results and thus requires higher mesh resolution in regions where field fields are of interest rapidly

Meshing in Maxwell

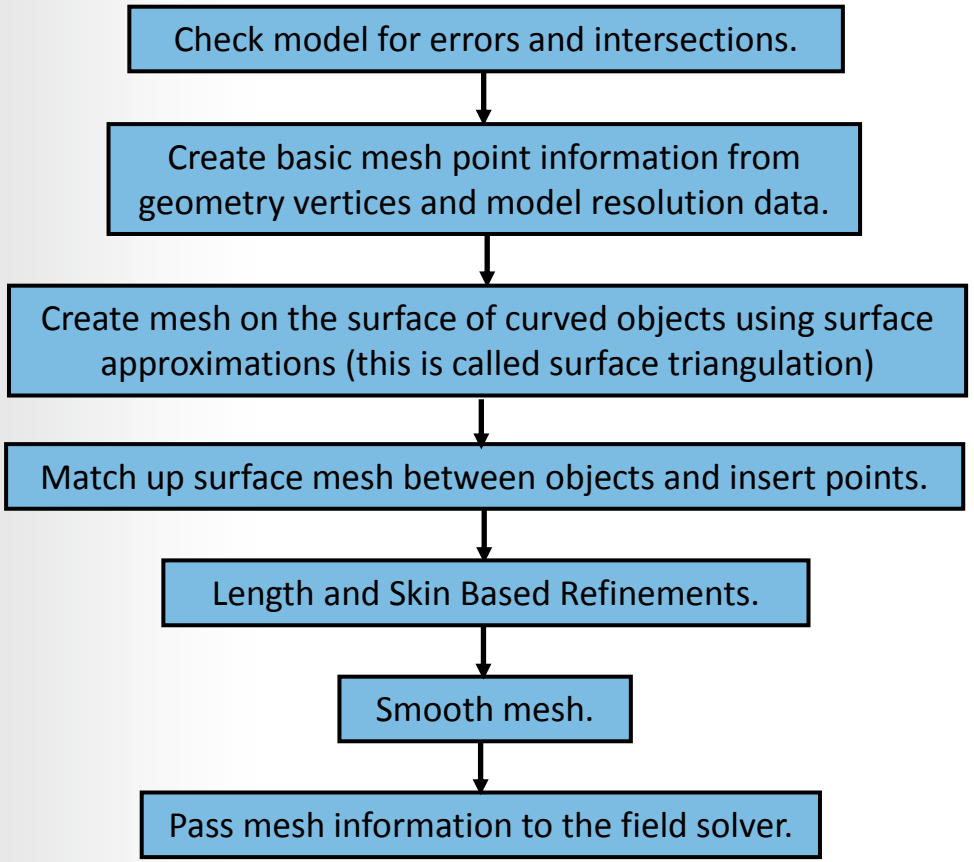
- Maxwell meshes all solids (model Objects) in the geometry automatically before solution process is started.
- In Maxwell's Static Solvers, the mesh is automatically refined to achieve the required level of accuracy in field computation. This is referred as Adaptive mesh refinement
- Maxwell also offers wide range of mesh operations which can be utilized to achieve a mesh as required by users

B. Initial Mesh

Initial Mesh

- When the Solution process is initiated, Maxwell uses an initial mesh to perform field calculations
- Initial mesh is automatically created by Maxwell without any instructions from users prior to performing field calculations

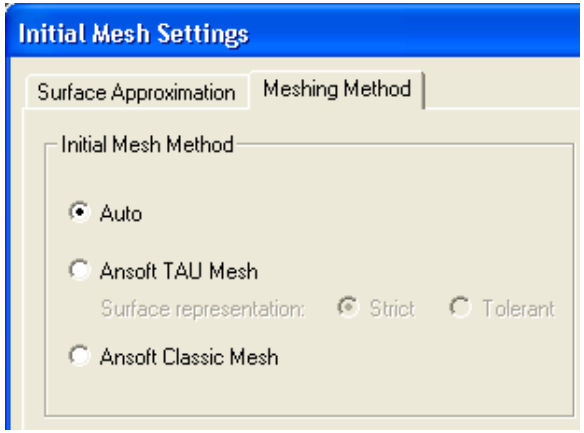
Initial Meshing Process



…Initial Mesh Settings

Initial Mesh Settings

- Default Initial Mesh Settings are appropriate for most geometries
- Initial Mesh settings can be accessed from the menu item ***Maxwell 3D → Mesh Operations → Initial Mesh Settings***



Meshing Methods

Auto (3D Only):

- This is default meshing method for Maxwell 3D
- Allows Maxwell to automatically select the appropriate mesher based on geometry

Ansoft Tau Mesh:

- Includes surface representation choices for Strict or Tolerant
- Strict algorithm tries to resolve curved surfaces more accurately while Tolerant algorithm uses loose tolerance for surface resolution
- For Complex and dirty geometries, Strict Mesher might fail while Tolerant mesher can create a mesh

Ansoft Classic Mesh:

- This is based on Ansoft 11 mesher
- Might not be suitable for Curved surfaces and requires geometry segmentation but can work better for Thin, Flat objects

Note: Options on Surface Approximation tab are same as Surface Approximation Mesh Operation which will be discussed later in this document

C. Adaptive Meshing

Adaptive Meshing

- For most of the cases, initial mesh is very coarse and more or less uniform in size throughout the region
- To achieve required level of accuracy in results, this mesh needs to be refined in areas where fields are of interest or the field gradients are high
- Adaptive meshing provides automated mesh refinement capability based on reported energy error in simulation
- Adaptive meshing is available only with static solvers

Solutions: puck_attractor - MaxwellDesign1

Design Variation: move='-1.5mm'

Simulation: Setup1

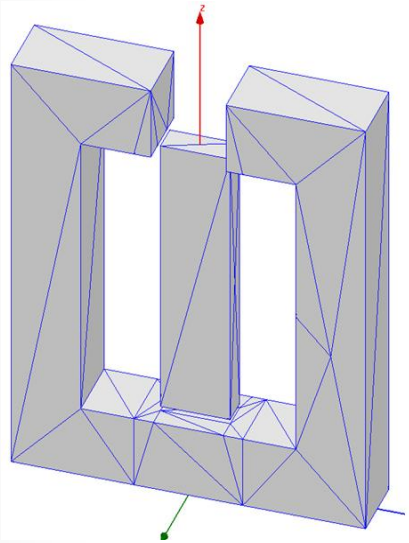
Solution | Convergence | Profile | Mesh Statistics

Number of Passes
Completed 22
Maximum 50
Minimum 2

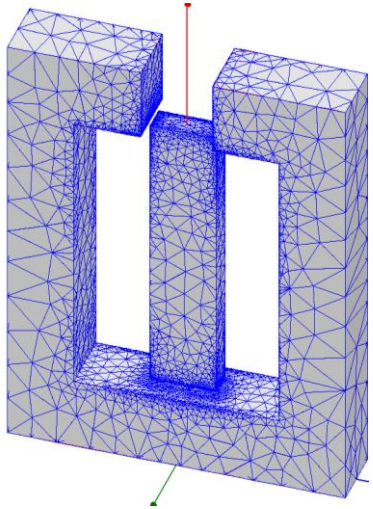
Energy Error/Delta Energy (%)
Target (0.01, 0.01)
Current (0.028747, 0.0014608)

View: Table Plot

Pass	Tets	Total Energy (J)	Delta Energy (%)	Energy Error (%)	Mag(F)/T	Output Var.	Output Var. Delta (%)
1	444	0.0089591	N/A	31.168	2.2015	2.2015	N/A
2	597	0.0091952	2.5677	13.088	1.9253	1.9253	14.349
3	788	0.0096087	4.3037	9.8546	2.0655	2.0655	6.7897
4	1056	0.0098166	2.1184	6.8009	2.0557	2.0557	0.47512
5	1400	0.0099763	1.5999	4.3254	2.1551	2.1551	4.6128
6	1828	0.0099927	0.16444	3.0868	2.1365	2.1365	0.87111
7	2387	0.010021	0.28024	2.1984	2.1556	2.1556	0.88311
8	3130	0.010039	0.17903	1.5954	2.1545	2.1545	0.051205
9	4067	0.010041	0.019227	1.1807	2.1593	2.1593	0.22495
10	5250	0.010044	0.028341	0.93393	2.1579	2.1579	0.066248
11	6772	0.010047	0.036796	0.6828	2.1608	2.1608	0.13363
12	8666	0.010049	0.016312	0.51382	2.159	2.159	0.083232
13	11188	0.010051	0.017316	0.37704	2.1614	2.1614	0.11189
14	14298	0.010053	0.025576	0.28337	2.1641	2.1641	0.12709
15	18201	0.010056	0.031492	0.2148	2.1626	2.1626	0.073287
16	23216	0.010059	0.029952	0.1635	2.1632	2.1632	0.031387
17	29606	0.010062	0.028738	0.11957	2.1632	2.1632	0.00023554
18	37758	0.010064	0.015231	0.091903	2.1632	2.1632	0.0023981
19	48059	0.010065	0.0092741	0.067301	2.1639	2.1639	0.031002
20	61270	0.010065	0.0027072	0.050487	2.164	2.164	0.0044363
21	78086	0.010065	0.00092893	0.038198	2.1641	2.1641	0.0048059
22	99366	0.010065	0.0014608	0.028747	2.1642	2.1642	0.007855



Initial Mesh

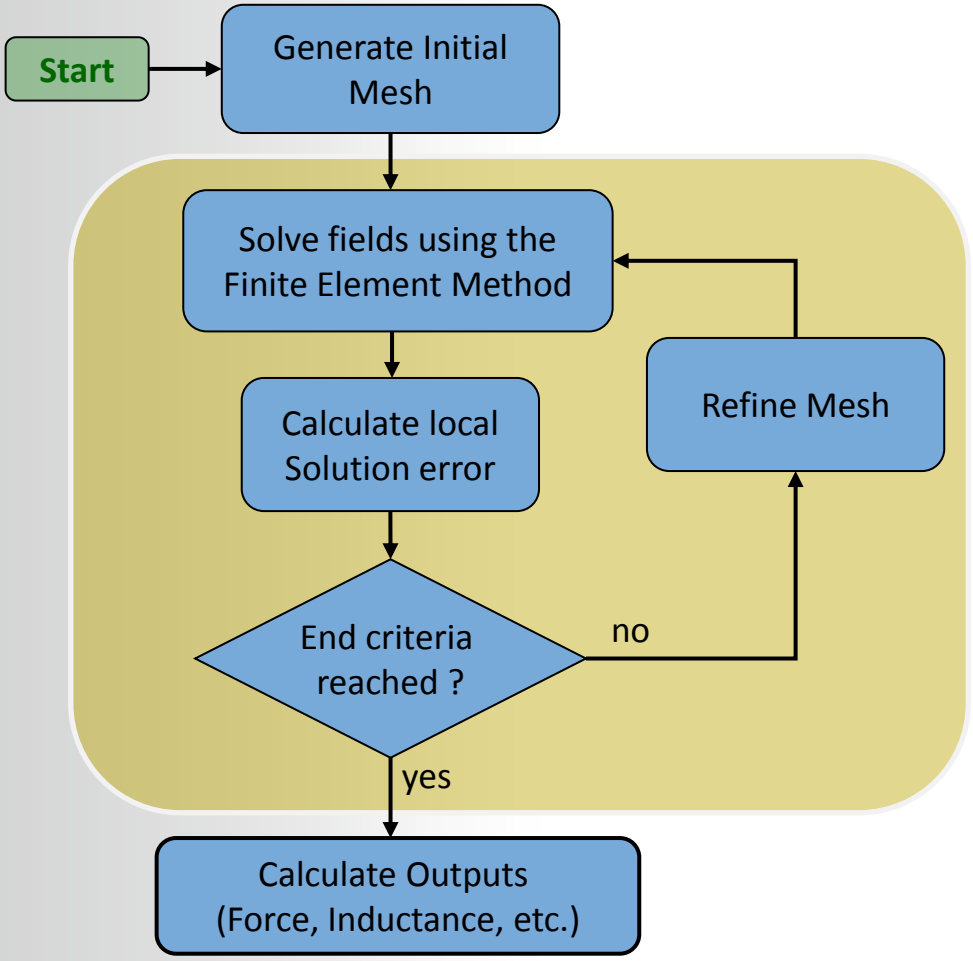


Adaptively Refined Mesh

…Adaptive Meshing

Adaptive Meshing Workflow

– Adaptive meshing technique start with initial mesh and refines it until required accuracy is met or Maximum number of passes is reached



D. Mesh Operations

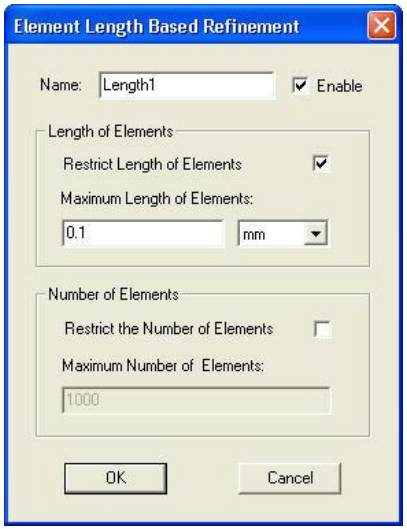
Mesh Operations

- Maxwell's Adaptive mesh refinement feature can be effectively used to achieve an optimized mesh for static solvers
- Transient Solvers does not have this capability to improve the initial mesh. Thus Transient Solvers require either Mesh Operations to be specified, or use the Link Mesh option to an adaptively refined mesh from a static solver.
- In Complex Static problems, it is also recommended to use Mesh Operations
  - To reduce number of passes required to achieve desired accuracy
  - To increase mesh density in areas of interest before the adaptive mesh refinement solution begins.
- Maxwell 16 offers following mesh operation specifications
  - On Selection/ Length Based;
  - On Selection / Skin Depth Based
  - Inside Selection / Length Based
  - Surface Approximation
  - Model Resolution
  - Cylindrical Gap Treatment

a. On Selection Mesh Operation

Mesh Operation: On Selection/Length Based

- The Length-based On-selection refinement will limit the edge length of all triangles formed on the surface of a selected object or any selected faces.
- This mesh operation can be added from the menu item ***Maxwell 2D/3D → Mesh Operations → Assign → On Selection → Length Based***



Restrict Length of Elements:

- Refines the mesh by controlling maximum size of the elements on the boundary of assigned object

Restrict Length of Elements:

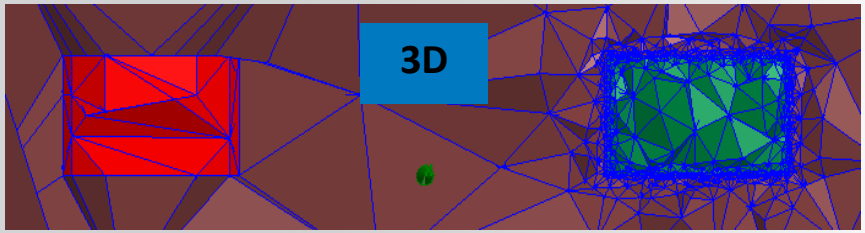
- Sets Maximum length of the elements that assigned object can have

Restrict the Number of Elements:

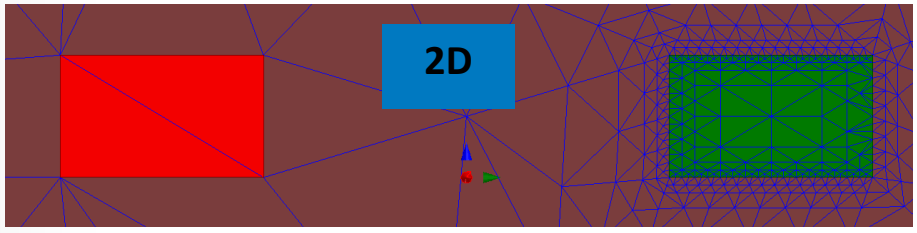
- Refines the mesh by controlling maximum element count on the boundary of assigned object

Maximum Number of Elements:

- Sets Maximum element count on the assigned object



Without Mesh Operation | With Mesh Operation



Without Mesh Operation | With Mesh Operation

Note: When Restrict Length of Elements and Restrict Number of Elements both are selected, mesh refinement will stop when any of the conditions are met

…On Selection Mesh Operation

Mesh Operation: On Selection/Skin Depth Based

- Skin Depth Based mesh operations are assigned to resolve induced eddy current near the surface of the conductor
- This refinement method creates layers of mesh within the selected surfaces of objects
- Skin depth based mesh operation can be assigned from ***Maxwell 2D/3D → Mesh Operations → Assign → On Selection → Skin Depth Based***

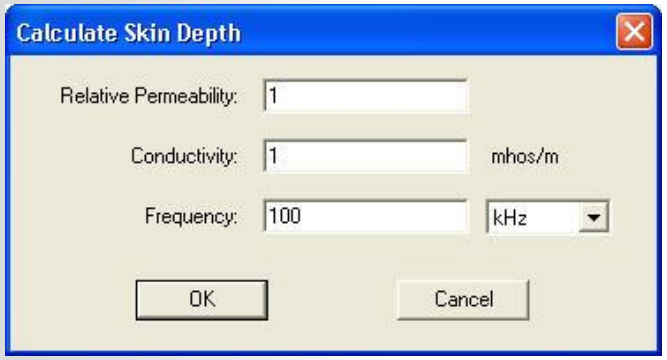
Skin Depth:

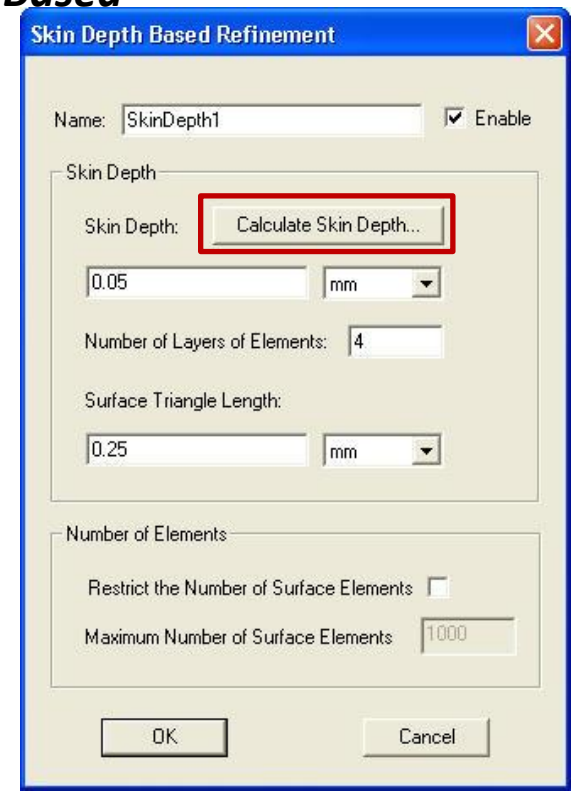
- Skin Depth field allows users to enter known value of the skin depth and number of layers of mesh to be created

Calculate Skin Depth:

- Calculate Skin Depth tab allows user to compute resulting skin depth value based on entered Permeability, Conductivity and Frequency
- Computed value is automatically assigned in Skin Depth field

$$\delta = \sqrt{\frac{2}{\omega\sigma\mu_0\mu_r}} = \frac{1}{\sqrt{\pi f\sigma\mu_0\mu_r}}$$





…On Selection Mesh Operation

Mesh Operation: On Selection/Skin Depth Based *(Contd…)*

Number of Layers of Elements:

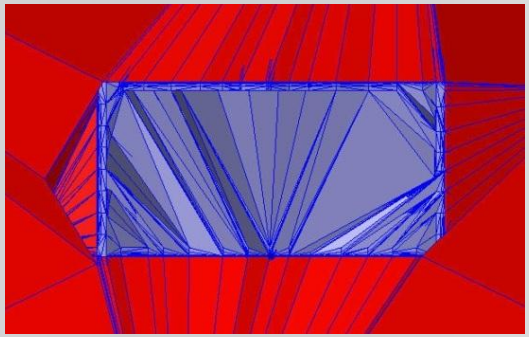
- Sets maximum number of mesh layers created in skin region

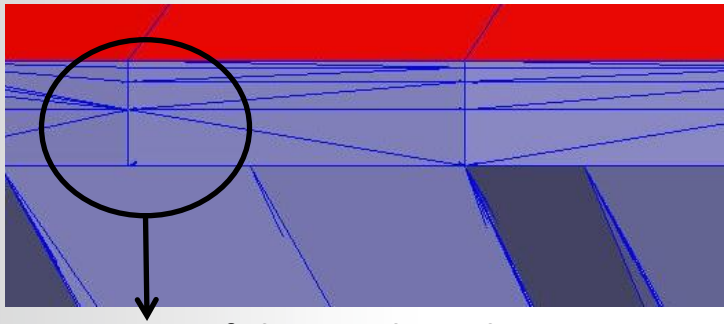
Surface Triangle Length:

- Sets the maximum size of elements on the assigned objects
- Surface Triangle Length controls the aspect ratio of the elements in skin depth region

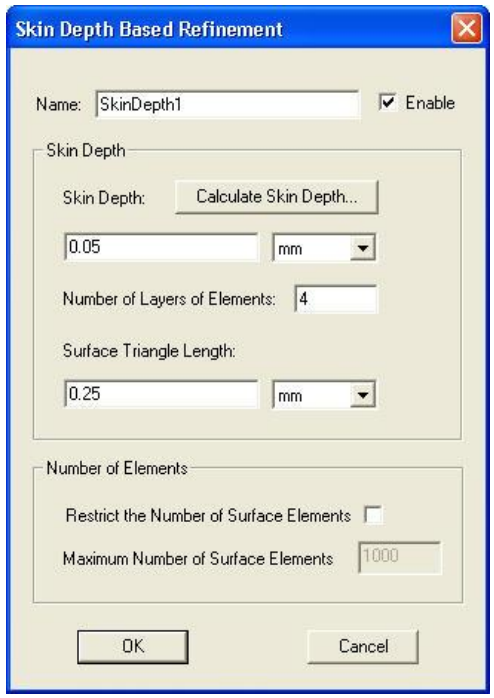
Restrict the Number of Surface Elements:

- Restricts the count of elements to the specified value on surface of assigned object





Four Layers of Skin Depth Mesh

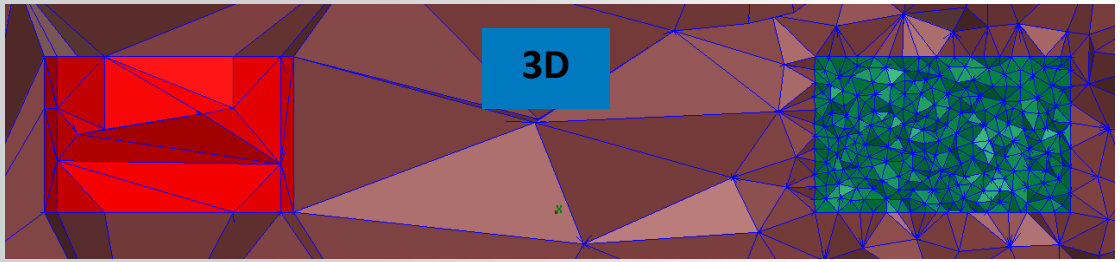


Note: Skin Depth Based mesh operation may result in high aspect ration tetrahedrons, thus it should be used very carefully

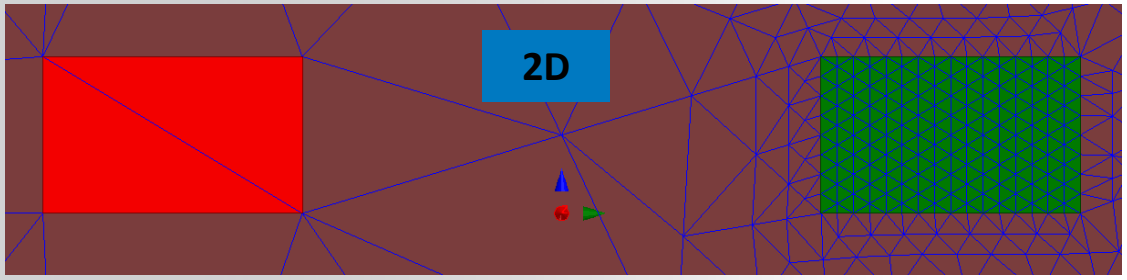
b. Inside Selection Mesh Operation

Mesh Operation: Inside Selection/Length Based

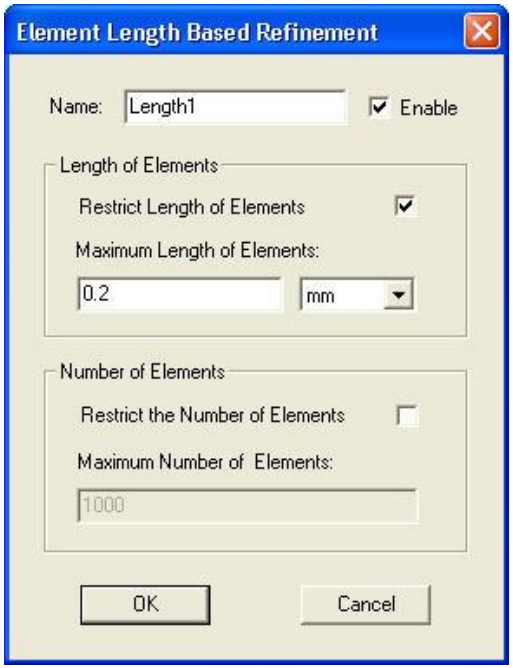
- The Length-based Inside-selection refinement will limit the edge length of all tetrahedrons (or triangles) formed inside a selected solid or sheet object
- This mesh operation can be added from the menu item ***Maxwell 2D/3D → Mesh Operations → Assign → Inside Selection → Length Based***
- All the options in the Element Length Based Refinement window are the same as for On Selection mesh operation except that the inside selection refinement will control size or number of elements inside the selected object



Without Mesh Operation **With Mesh Operation**



Without Mesh Operation **With Mesh Operation**



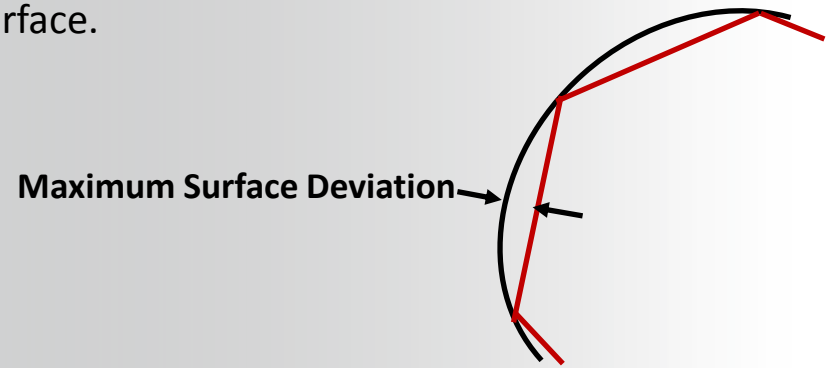
c. Surface Approximation

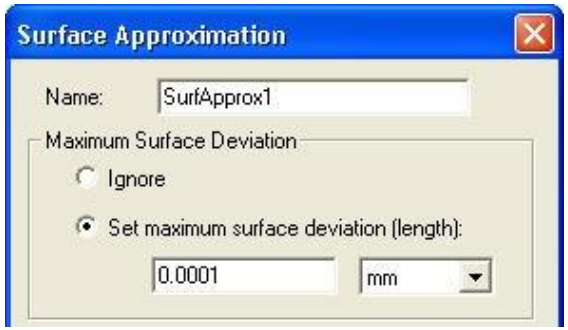
Surface Approximation

- Surface Approximation Mesh Operations are helpful to resolve curved surfaces of the geometry with a good quality mesh and can be used to both increase or decrease mesh density on curved surfaces
- By default, Surface Approximation mesh operation is performed while creating initial mesh for which parameters are set through Initial Mesh Settings
- For complex parts of the geometry, addition surface approximation can be assigned from ***Maxwell 3D → Mesh Operations → Assign → Surface Approximation***
- Note that Surface Approximation assignment or altering its parameters will remove the existing initial mesh

Maximum Surface Deviation:

- It is the maximum spacing, in drawing units, that the tetrahedral surfaces may be from the true-curved geometry's surface.



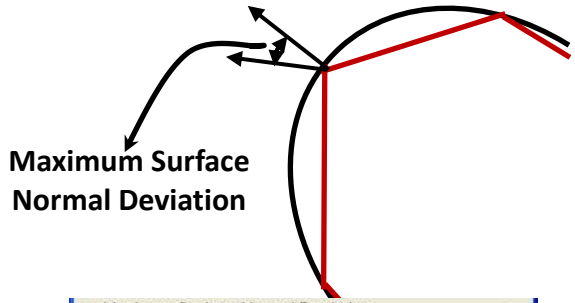


...Surface Approximation

Surface Approximation *(Contd...)*

Maximum Surface Normal Deviation:

- The maximum angular difference, in degrees, that a tetrahedral face's normal can have from the surface normal for the true geometry which it is meant to represent.
- The default value is 15 deg.

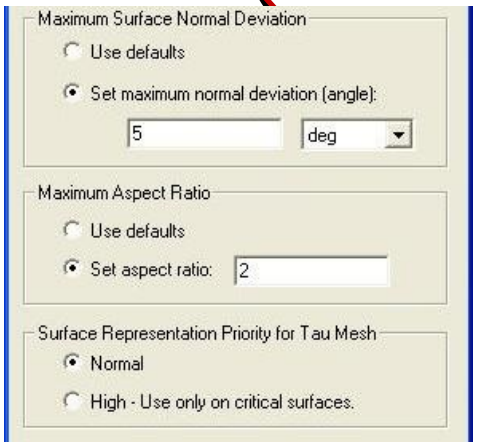


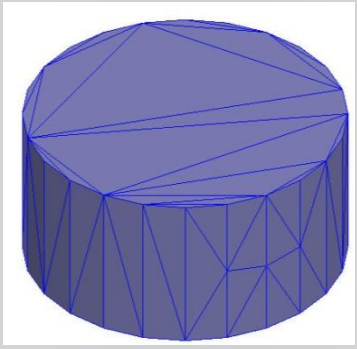
Maximum Aspect Ratio:

- The maximum allowed aspect ratio of all faces of all tetrahedral of the selected object or face. This setting influences mesh quality by limiting aspect ratio of resulting elements

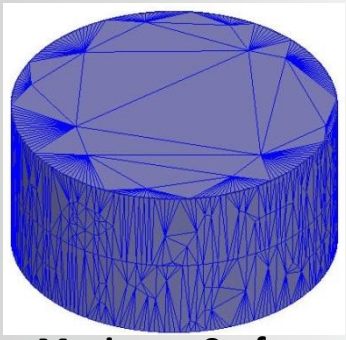
Surface Representation Priority for Tau Mesh

- In most cases, meshing is done by Tau Mesh. You can set the surface representation as normal or high

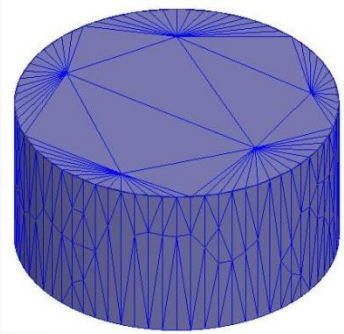




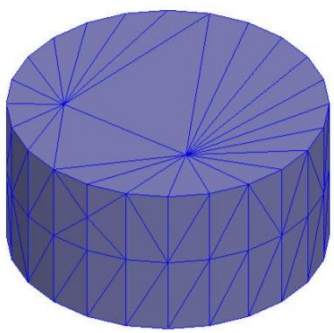
Default Mesh



Maximum Surface Deviation



Maximum Surface Normal Deviation

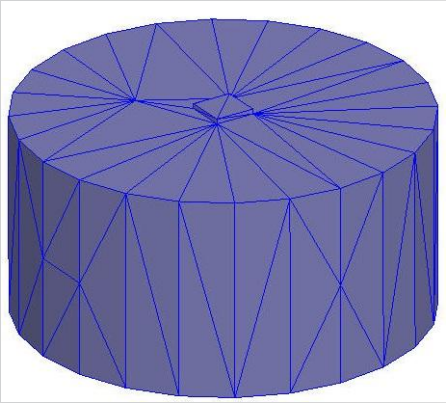


Maximum Aspect Ratio

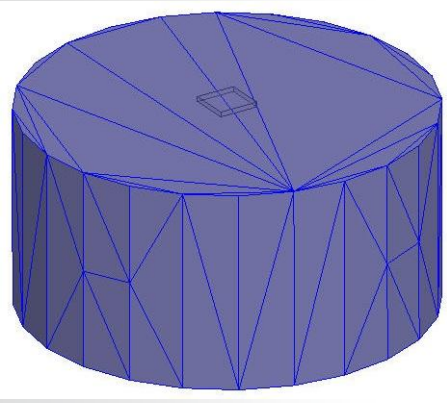
d. Model Resolution

Model Resolution

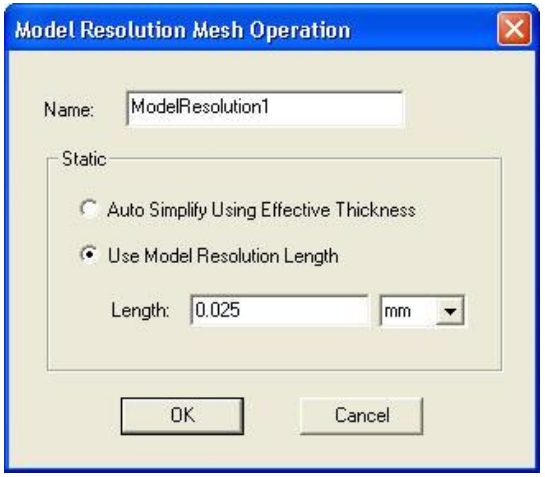
- Model Resolution enables users to ignore small features of geometry which might not be important from simulation point of view
- Users can specify the minimum length of geometry features which will be resolved by mesh and any feature below the specified size will be ignored
- Default Option is set to Auto Simplify which will automatically calculate the minimum feature length based on effective thickness of the object
- Mesh Operation can be assigned from menu item ***Maxwell 3D → Mesh Operations → Assign → Model Resolution***



Without Mesh Operation



With Mesh Resolution

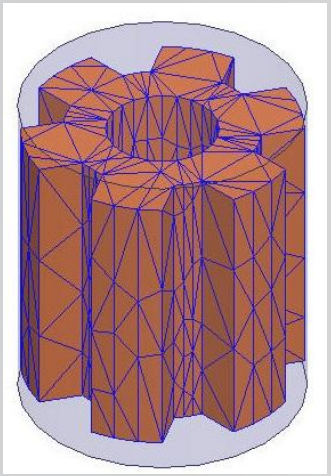


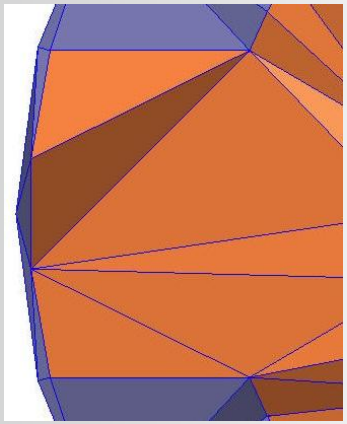
Note: Model resolution must be used with caution as sometimes mesh might not be able to represent geometry correctly

e. Cylindrical Gap Treatment

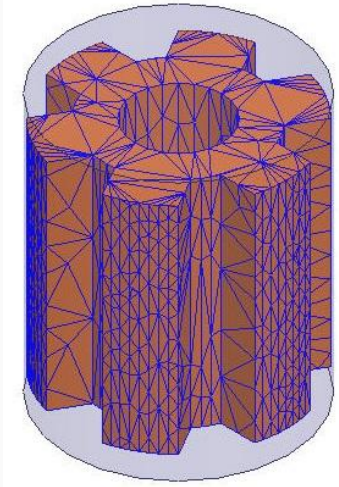
Cylindrical Gap Treatment

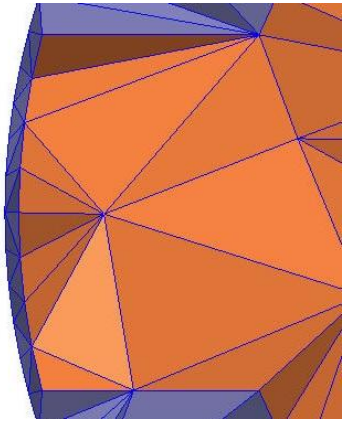
- Cylindrical Gap Treatment mesh operation is a proximity based mesh refinement and usually assigned to Band objects for rotational motion
- The refinement of mesh is done on the applied objects based on the closeness of the geometry lying inside it
- For Transient Solver involving rotational motion, this mesh operation is automatically created once the rotational motion is defined in order to resolve air gap between Stator and rotor parts
- Mesh Operation can be assigned from menu item ***Maxwell 3D → Mesh Operations → Assign → Cylindrical gap Treatment***





Without Mesh Operation





With Mesh Operation

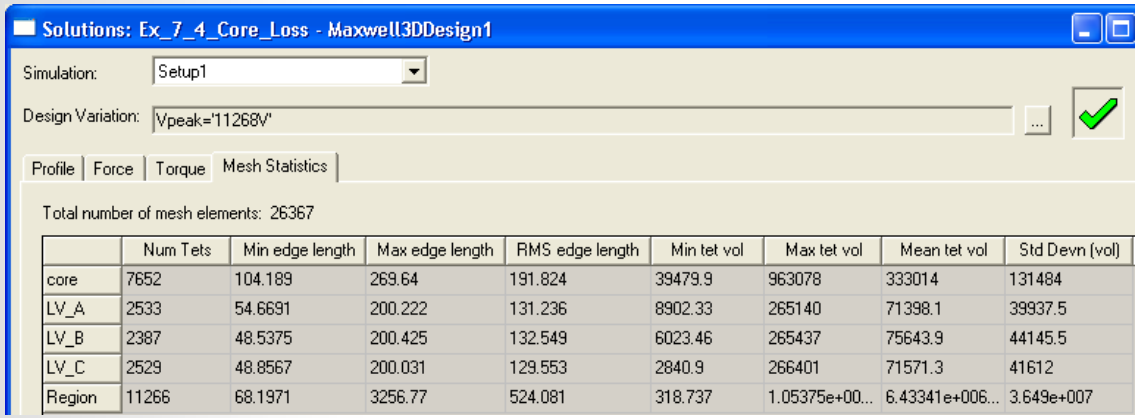
E. Applying Mesh Operations

Apply Mesh Operations

- When Analysis Process is started mesh operations are automatically applied on initial mesh
- It is advisable to assign mesh operations and verify mesh quality and element count before starting the solution process by inspecting both the Mesh Statistics, and visual inspection of Mesh plots.
- Mesh Operations can be assigned from menu item ***Maxwell 3D → Analysis Setup → Apply Mesh Operations*** or right click on Analysis Setup from Project Manager window and select **Apply Mesh Operations**

Mesh Statistics

- Once Mesh Operations are applied, mesh quality and element count can be verified from the ***Maxwell 3D → Results → Solution Data***
- In Solutions window, select Mesh Statistics tab

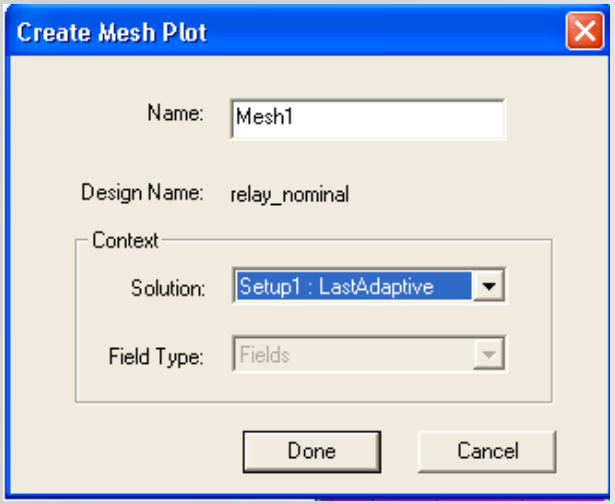


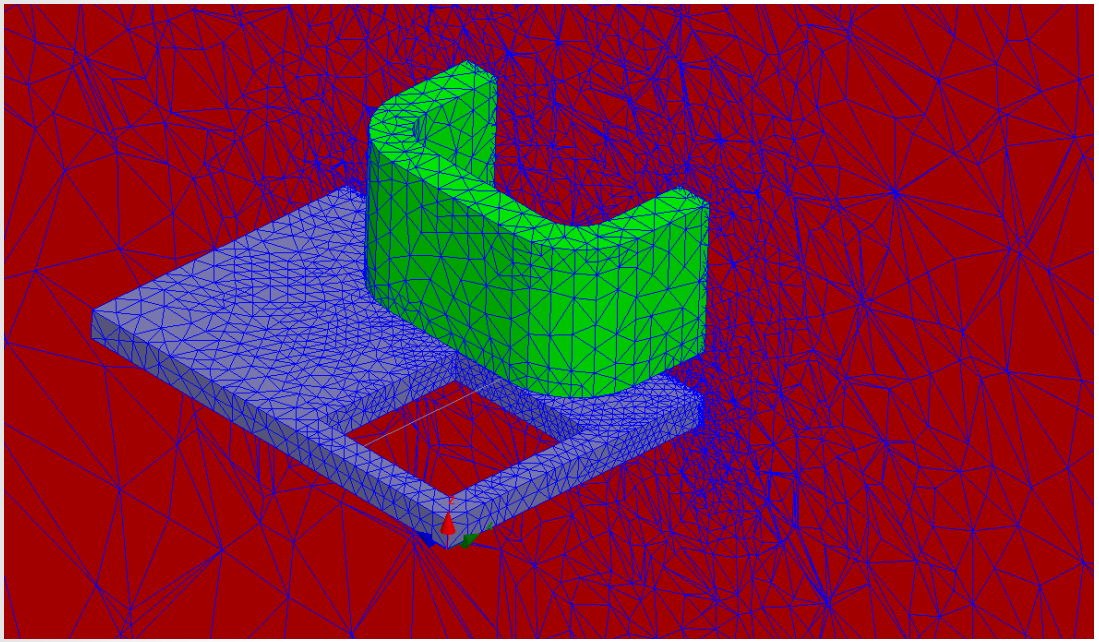
	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
core	7652	104.189	269.64	191.824	39479.9	963078	333014	131484
LV_A	2533	54.6691	200.222	131.236	8902.33	265140	71398.1	39937.5
LV_B	2387	48.5375	200.425	132.549	6023.46	265437	75643.9	44145.5
LV_C	2529	48.8567	200.031	129.553	2840.9	266401	71571.3	41612
Region	11266	68.1971	3256.77	524.081	318.737	1.05375e+00...	6.43341e+006...	3.649e+007

… Applying Mesh Operations

Mesh Plots

- Mesh plots enables to insect the mesh on objects or the sections of mesh to verify it validity
- A Mesh plot can be created on the objects, sheets or the planes
- To create the mesh plot, select the required entities and then select the menu item ***Maxwell 3D/2D → Fields → Plot Mesh***

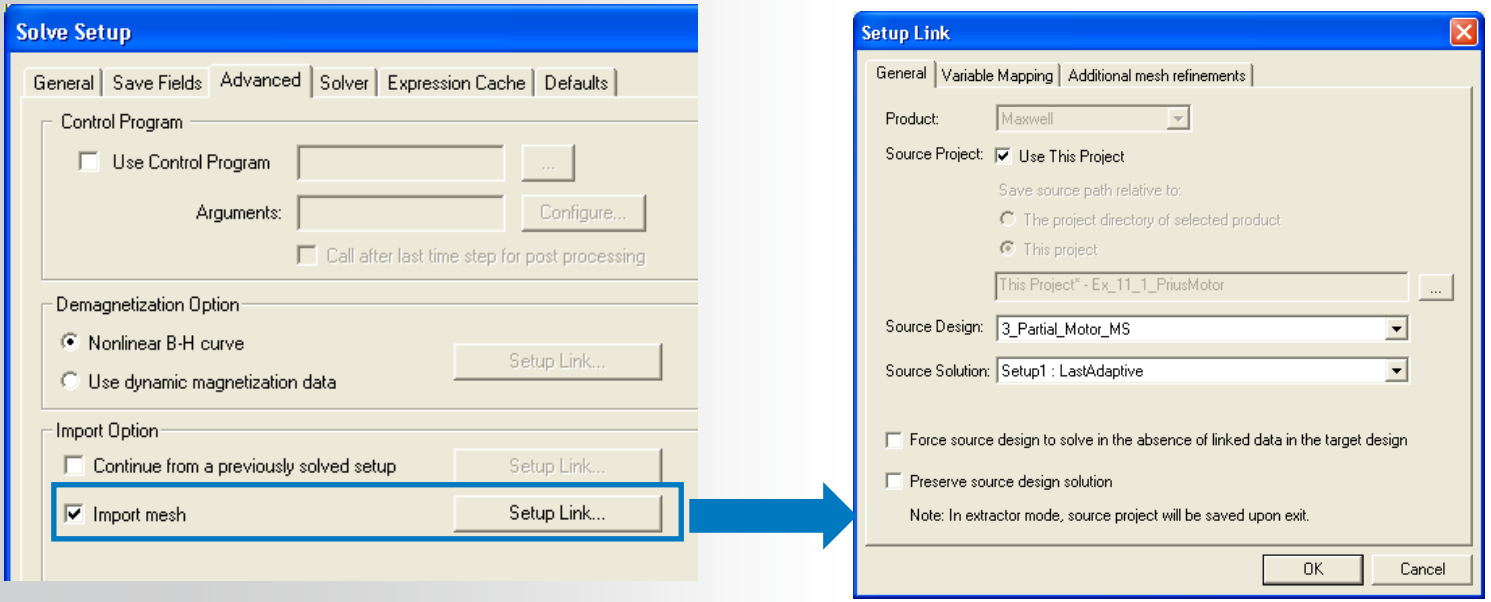




F. Mesh Linking

Linking Mesh to Other Design

- In some of the static cases as well it is beneficial to link meshes across the design to achieve optimum results
- A Transient can also link the an adaptively refined mesh from a Static Solution
- Mesh can be linked from Analysis Setup window
  - Import Mesh option is available under Solver tab for static solvers and Advanced tab for Transient Solver
- Note that Source and Target design should have exactly same geometry



G. Troubleshooting

Mesh Failure Troubleshooting

- Mesh generation might fail due to various reasons related to geometry
- If mesh failure occurs, users are advised to follow below steps
  - Select the menu item ***Modeler → Model Analysis > Show Analysis Dialog > Last Simulation Mesh*** to identify reason for mesh failure
  - Use the command ***Modeler → Model Analysis → Analyze Object*** to analyze geometry errors and perform healing
  - Use the command ***Modeler → Model Analysis → Analyze InterObject Misalignments*** to analyze and correct misalignments
  - Turn some parts of the geometry to Non-Model and perform meshing to identify exact problem region
  - Remove or simplify unnecessary complex features which are causing problem in meshing by redrawing
  - Use Surface Approximation for higher curvature objects to resolve curved faces
  - Use Model resolution cautiously to neglect unimportant small features