FEA Services

Heat Exchanger Finite Element Model (FEM) Methodology

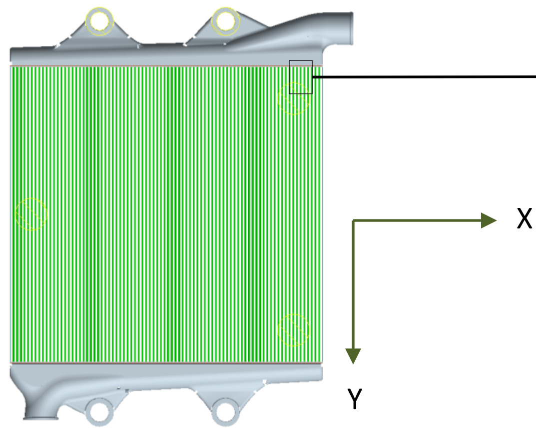
FEA Services LLC ©
15720 Brixham Hill Ave
Suite 300
Charlotte, NC 28277
704.594.1113

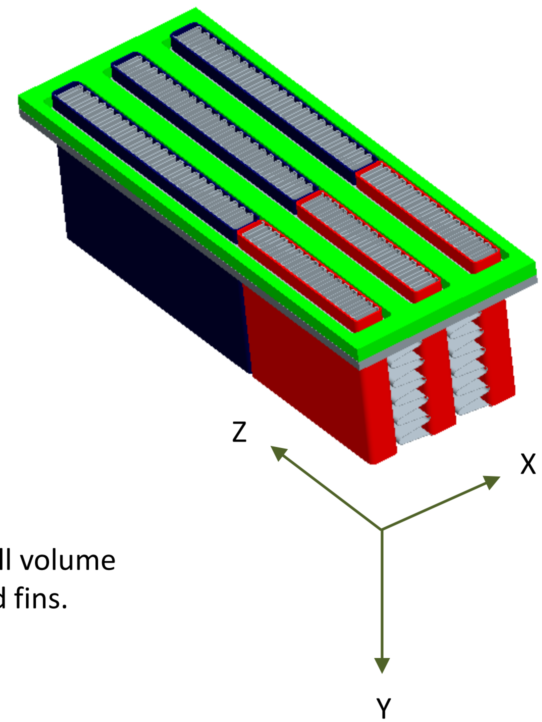
Summary

- Methodology has already been developed to make detailed fatigue analysis for complex heat exchanger (HE) applications yet with minimized effort. (i.e., “optimal” approach)
- Mechanical FEA driven by steady state temperature changes (dT) mapped over X-Y-Z coordinates. Often “Z” coordinate temperatures are considered constant.
- A larger “global” FEM of the entire application is built using anisotropic stiffness properties to represent the cooling fins.
- Global deflections and force reactions are recorded over the entire X-Y-Z volume of the HE.
- Localized, highly refined FEM is then built around the area of interest. Boundaries for this sub-model are taken from the global results. Stresses of interest are taken from the refined sub-model.
- FEA Services is also experienced with fatigue analysis and live real world test correlation with strain-life and damage tolerance methods thus can assist the customer with follow-on fatigue calculations or other assessments regarding fatigue implications.

Application

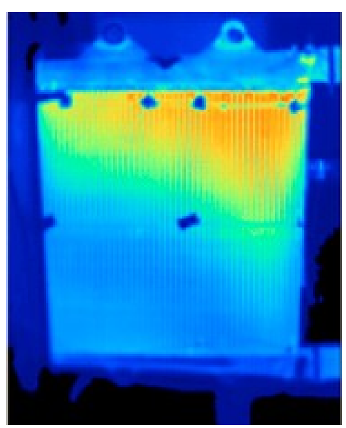
HE Geometry



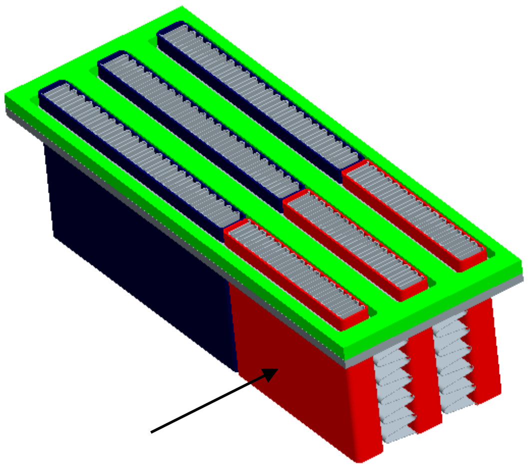


Consider a small volume of encapsulated fins.

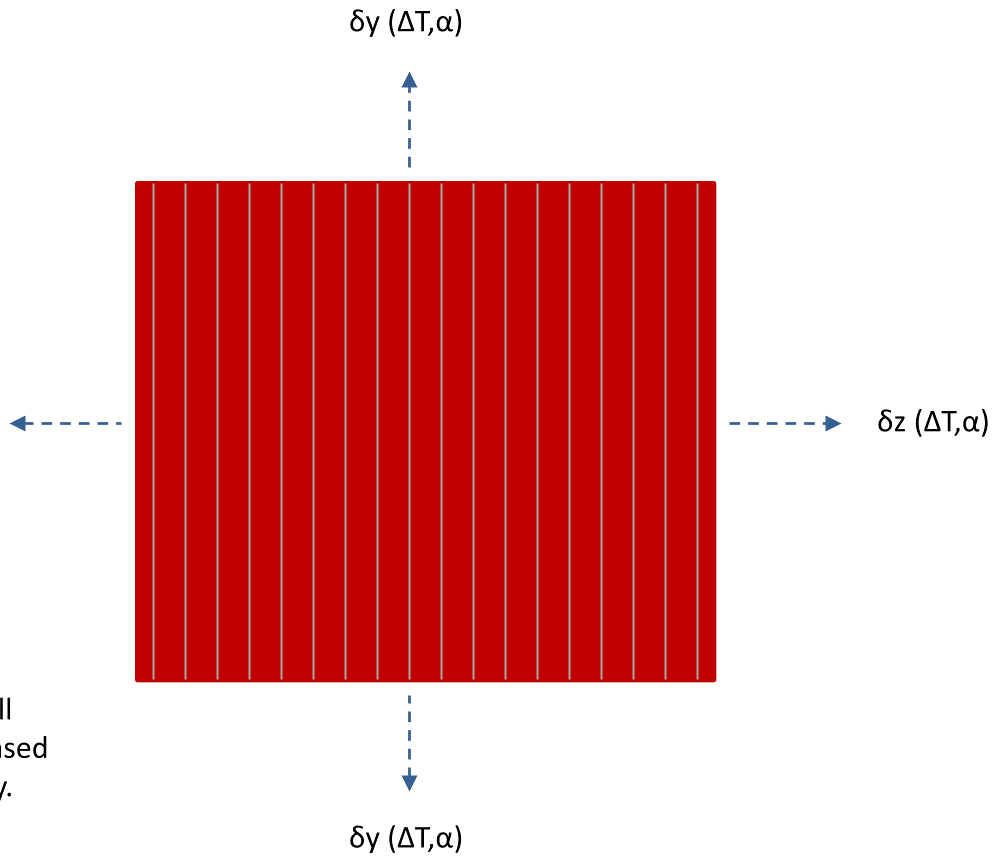
Typical Thermal Map



Global Anisotropic Elements

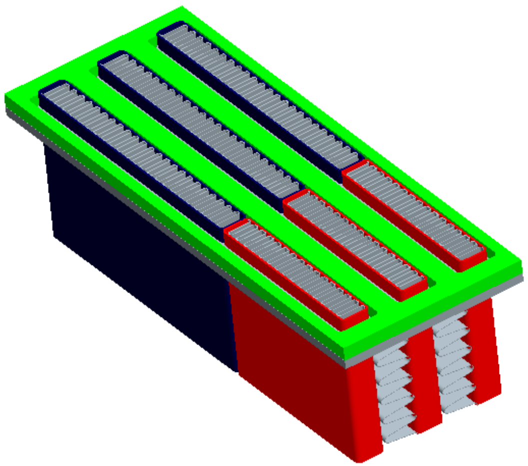


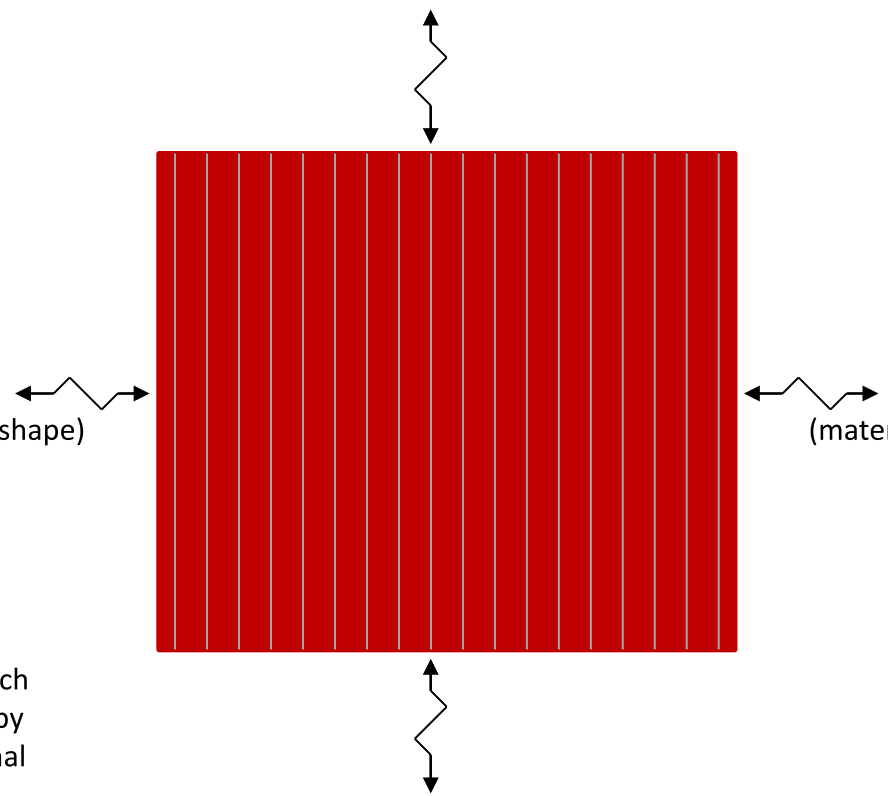
Look at the face of the red encapsulations with internal fins.



Free thermal growth in all directions due to dT is based on outer dimensions only.

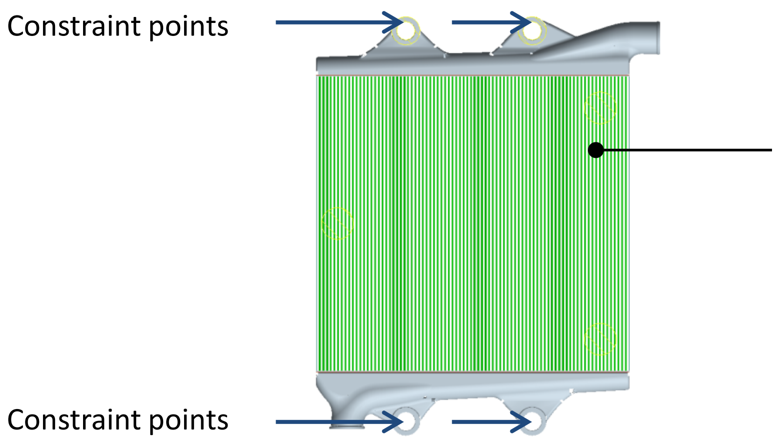
Global Anisotropic Elements





The STIFFNESS (E) in each direction is influenced by the design of the internal fins. (material, shape)

Global Anisotropic Elements



The STRESS at any point is based on the resistance to the free thermal growth.

The mechanical resistance to growth results in a net deflection at every point on the HE.

Ultimately the resistance is based either on dissimilar metals or the outer boundary constraints.

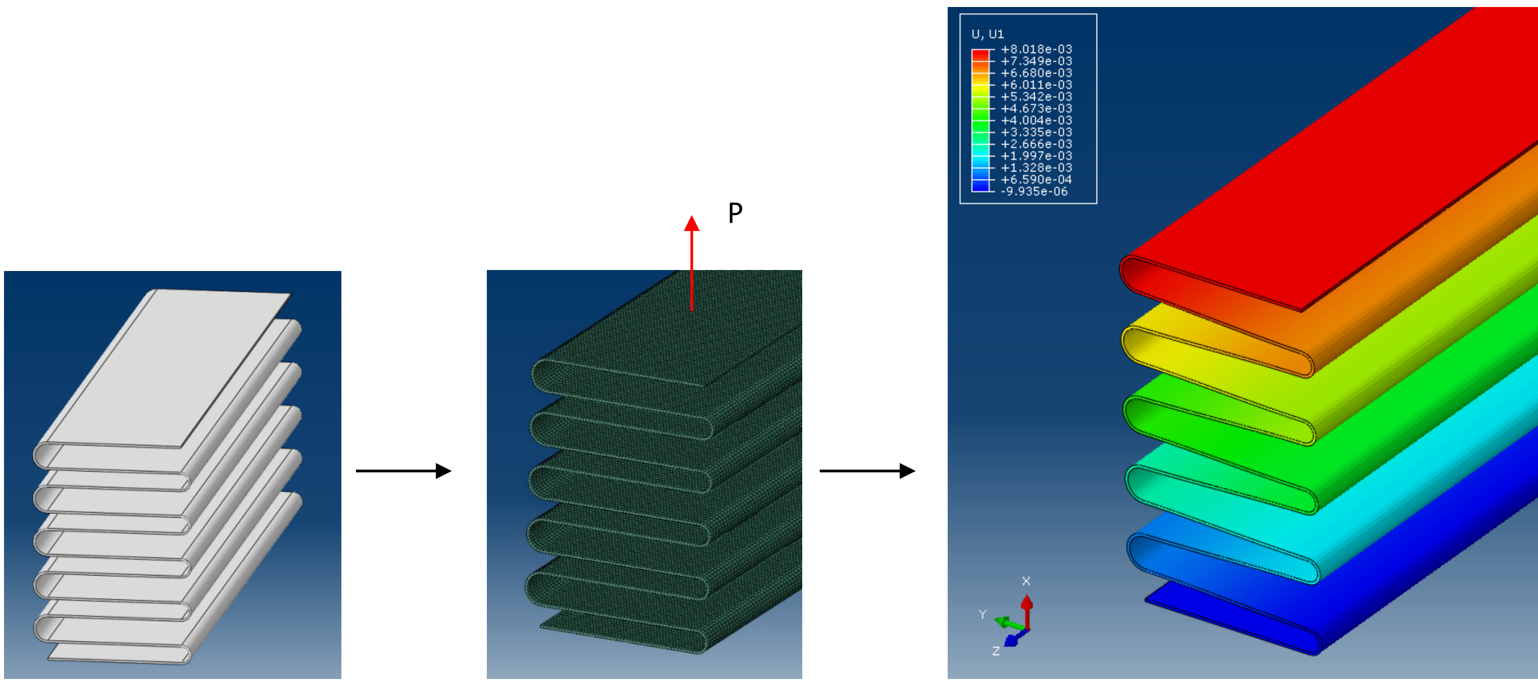
The purpose of the global FEA is to derive the net mechanical deflection at all points over the XYZ space.

Global Anisotropic Elements

Development of the HE FEA begins with developing an anisotropic 3D solid element which represents the stiffness of one row of fins:

1. Create a discrete FEM of a representative “block size” of fins.

2. Apply directional loads to the discrete FEM for the fin level FEA.



Global Anisotropic Elements

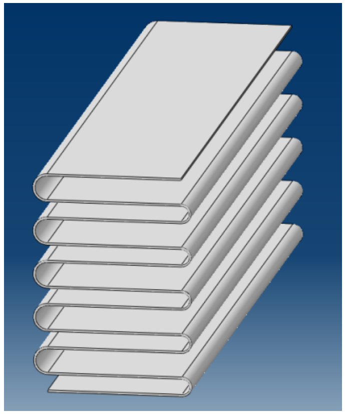
Development of the HE FEA begins with developing an anisotropic 3D solid element which represents the stiffness of one row of fins:

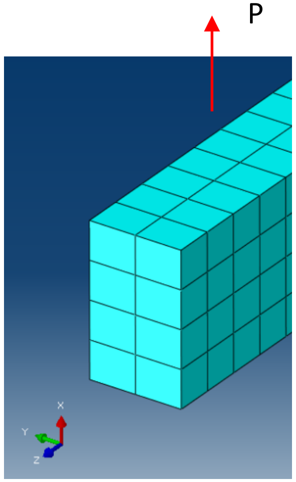
1. Create a discrete FEM of a representative “block size” of fins.

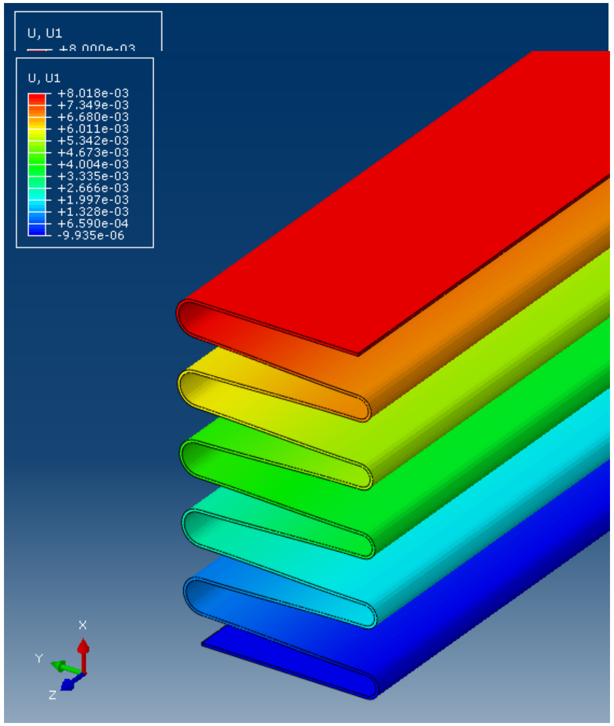
2. Apply directional loads to the discrete FEM for the fin level FEA.

3. Deflection results from this discrete FEA provides the necessary stiffness properties for the anisotropic solid elements.

4. The anisotropic elements are tested and verified with a small FEA.



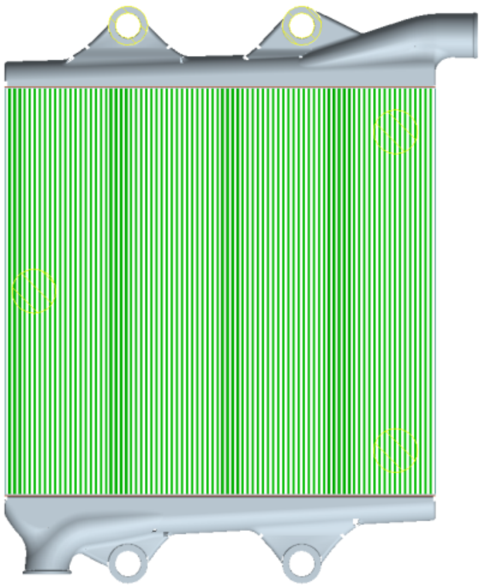




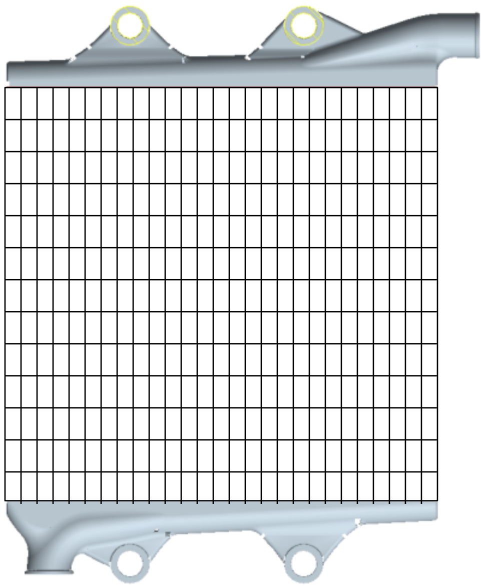
Global Anisotropic Elements

Development of the HE FEA begins with developing an anisotropic 3D solid element which represents the stiffness of one row of fins:

5. The anisotropic "block" elements completely represent the global behavior of the fins for the global HE FEA.



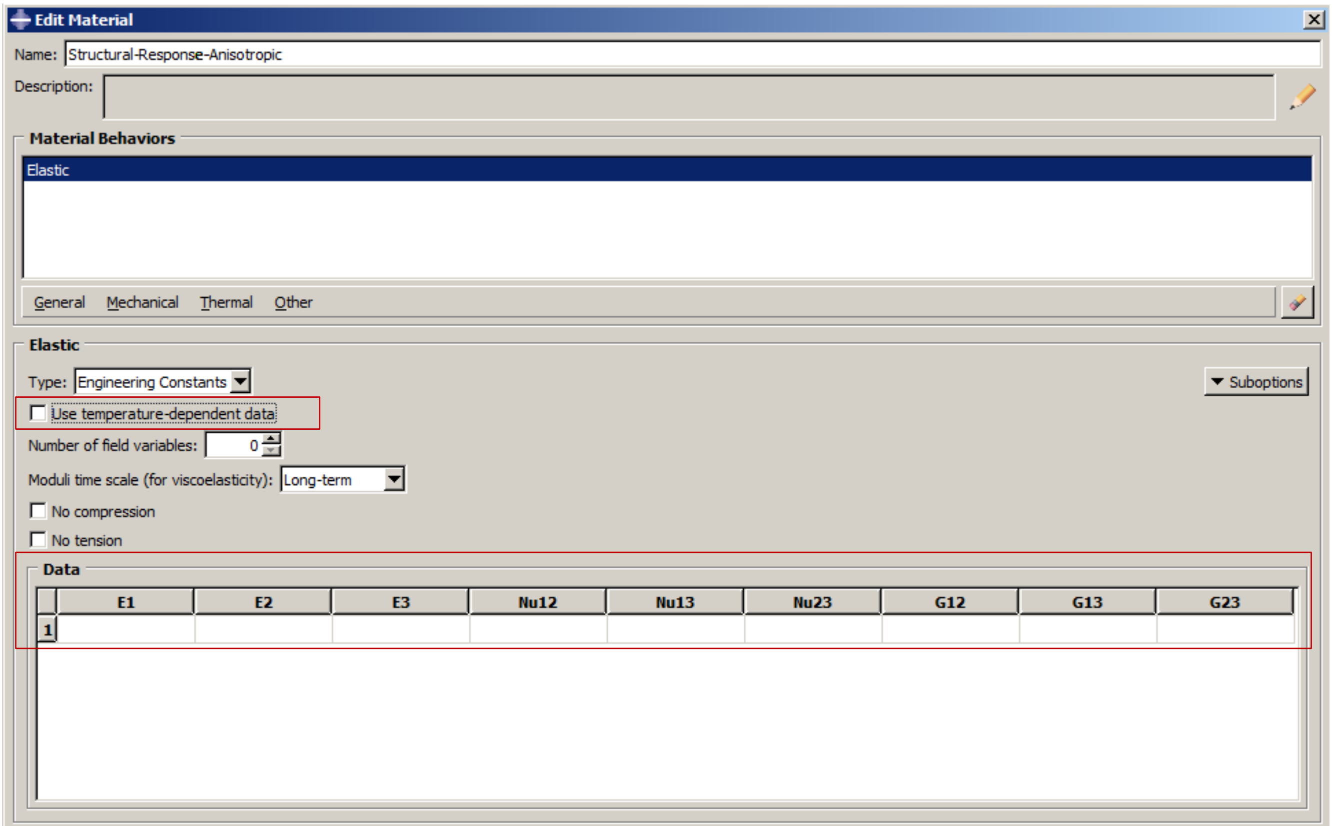
As-designed HE assembly



Representative global FEM

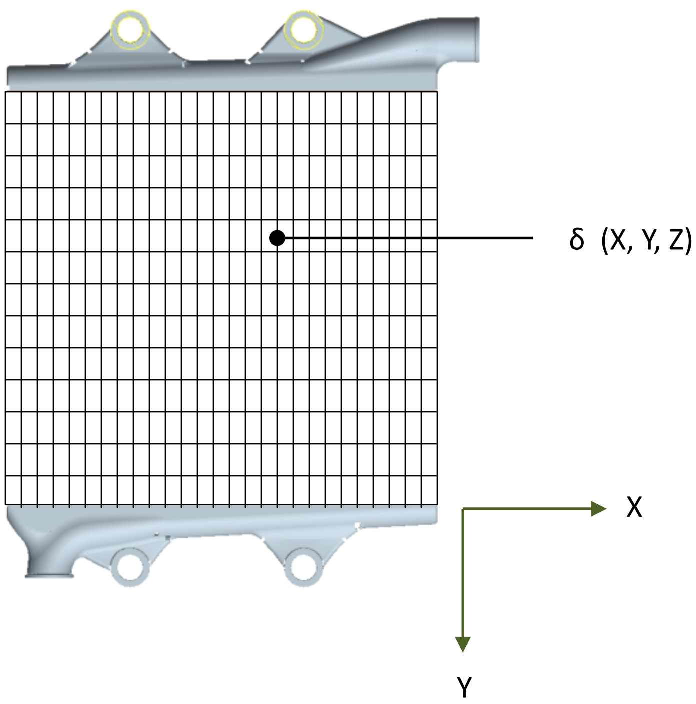
Global Anisotropic Elements

Inputs into the material model, as derived from the discrete FEM's:



Global Anisotropic Elements

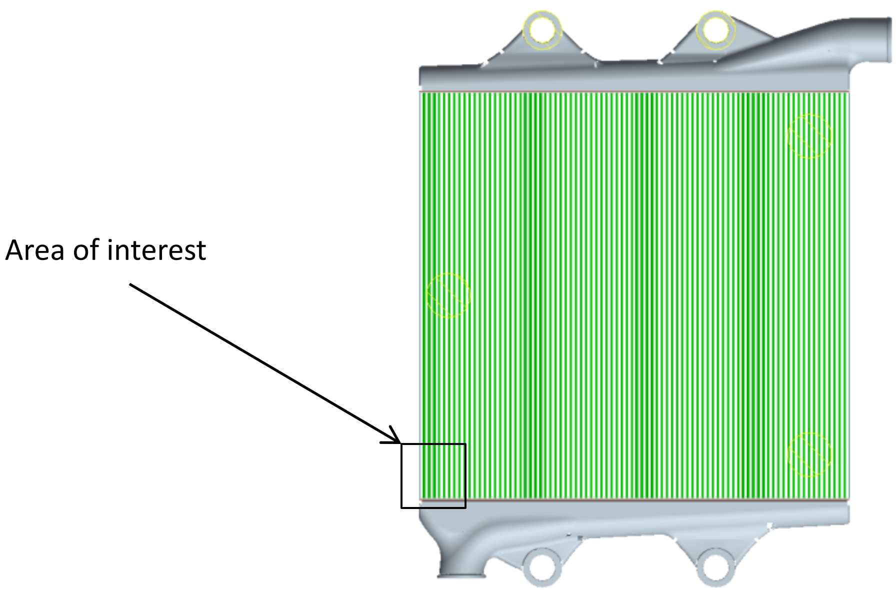
The GLOBAL FEM is run to produce **displacement results** throughout the structure.



Stress Analysis

Given the requirement to assess stress and/or fatigue in a small area, a more highly refined local model is created with high FE fidelity.

The displacement results which were derived by the global FEM become **boundaries** for the new local (sub) model, which is a highly refined 3D model in the area of interest.



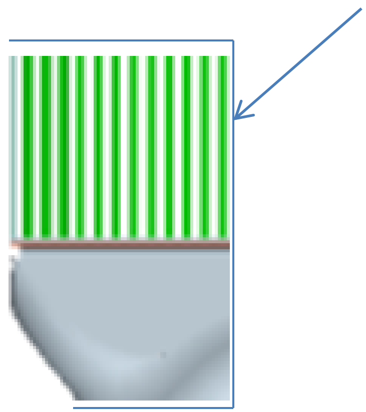
Stress Analysis

Given the requirement to assess stress and/or fatigue in a small area, a more highly refined local model is created with high FE fidelity.

The displacement results which were derived by the global FEM become **boundaries** for the new local (sub) model, which is a highly refined 3D model in the area of interest.

Boundary conditions taken automatically from the global model results

Area of interest



Intermediate Submodel