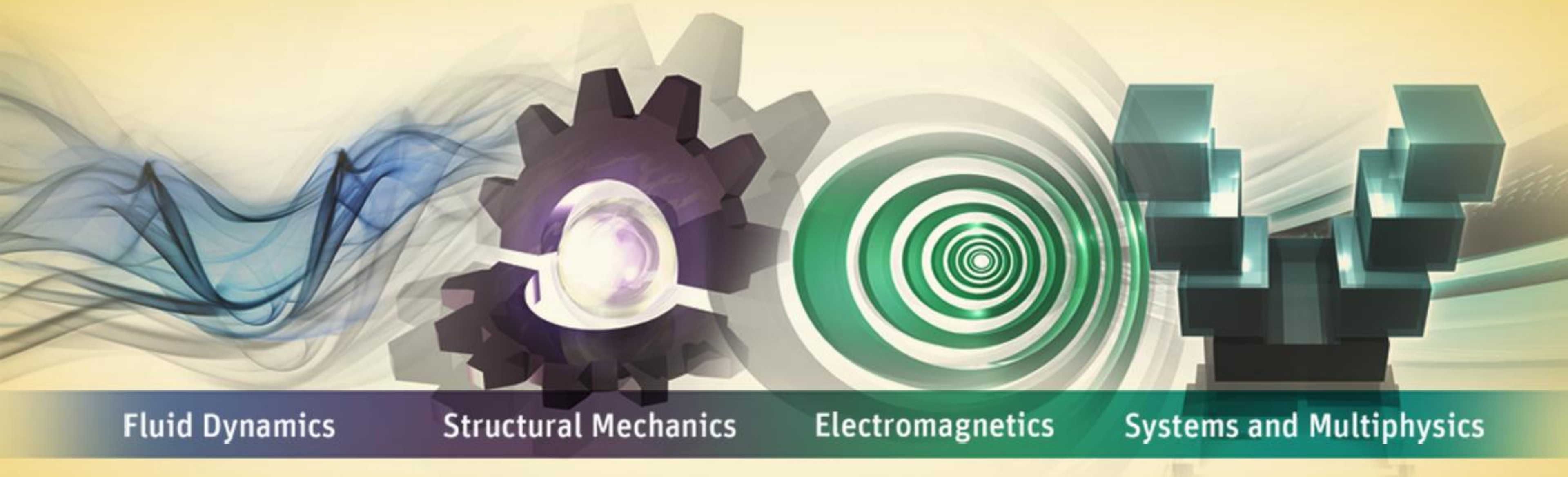
ANSYS

Realize Your Product Promise™

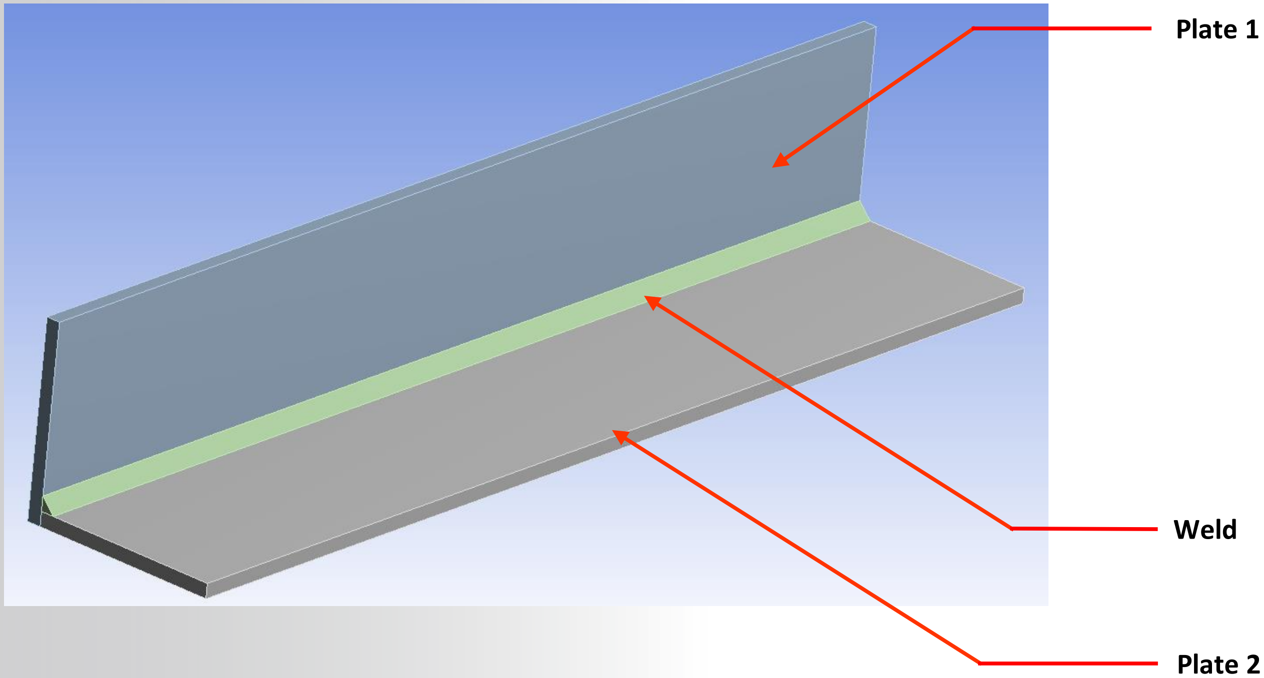
Simulation of Laser Welding Process



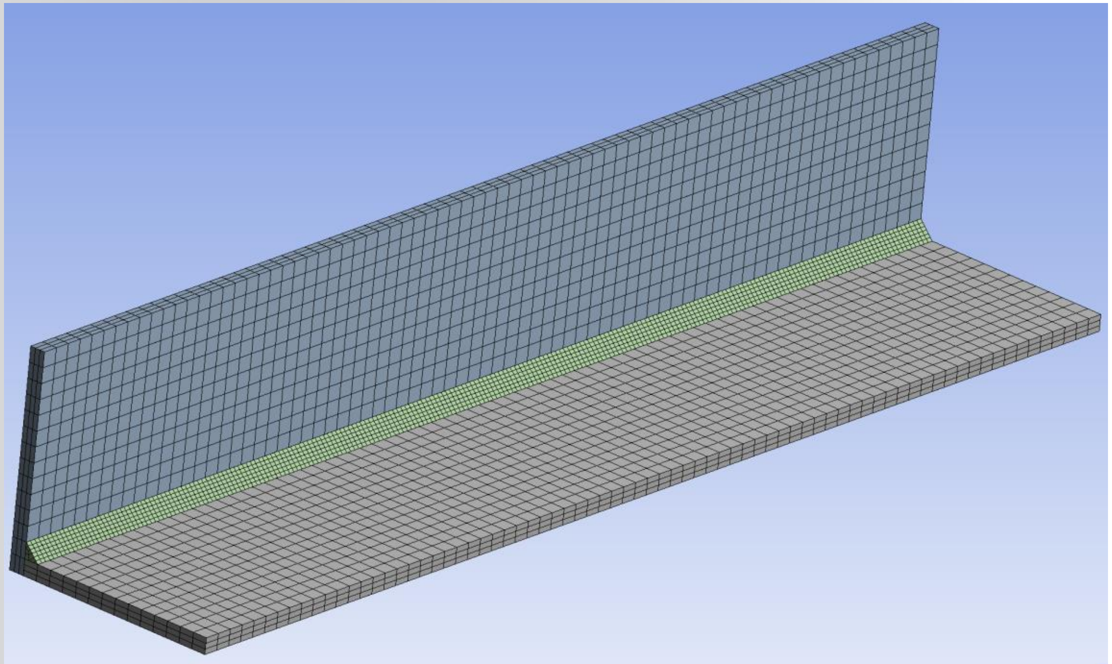
Ashutosh Srivastava

Problem Statement

Simulate the laser welding process to determine thermal stresses and heat affected zone in the structure. This is done on a simple model as shown below.



Mesh Details



Sweep mesh was used for the three components. For the plates, three elements were defined through the thickness to capture the bending effects. Lower order mesh was used in this case.

Component	Nodes	Elements
Plate 1	4560	3150
Plate 2	4560	3150
Weld	3094	1760
Total	12214	8060

Material Properties

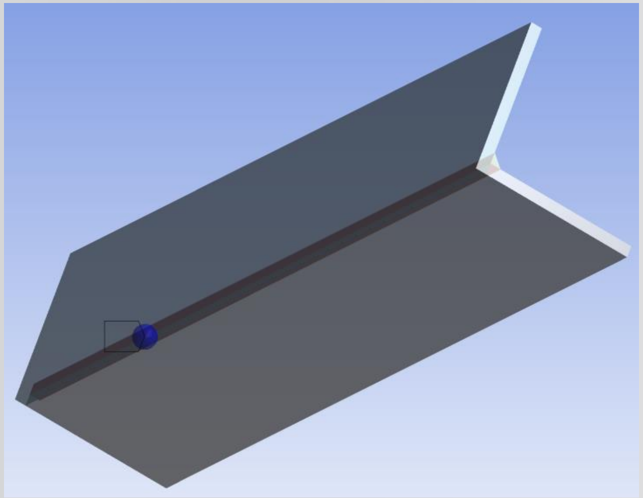
All the parts in the model were assigned same material properties as shown below. For simplicity, no temperature dependent properties were considered. Also material is considered to be linear elastic.

A	B	C
Property	Value	Unit
Density	7850	kg m^-3
Isotropic Secant Coefficient of Thermal Expansion		
Coefficient of Thermal Expansion	1.2E-05	C^-1
Reference Temperature	22	C
Isotropic Elasticity		
Derive from	Young's Modulus and Pois...	
Young's Modulus	2E+11	Pa
Poisson's Ratio	0.3	
Bulk Modulus	1.666666667E+11	Pa
Shear Modulus	7.692307692E+10	Pa
Alternating Stress Mean Stress	Tabular	
Strain-Life Parameters		
Tensile Yield Strength	250000000	Pa
Compressive Yield Strength	250000000	Pa
Tensile Ultimate Strength	460000000	Pa
Compressive Ultimate Strength	0	Pa
Isotropic Thermal Conductivity	60.5	W m^-1 C^-1
Specific Heat	434	J kg^-1 C^-1

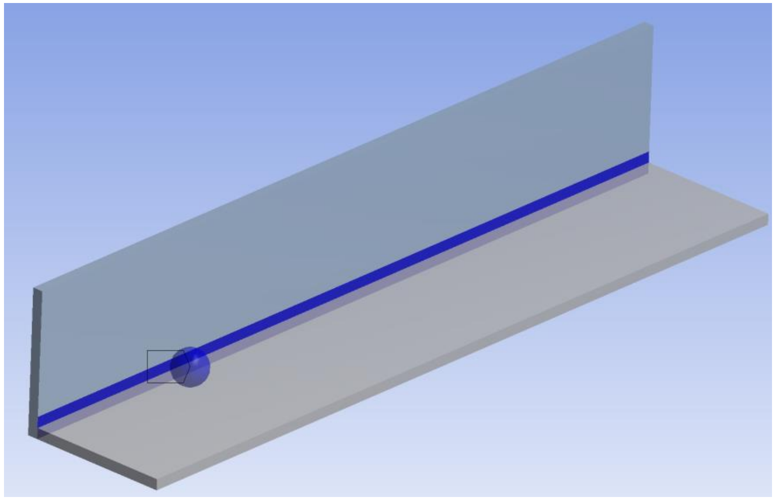
Contact Definitions

Two contact were made:

- Contact between the plates.
- Contacts between weld and the plates.

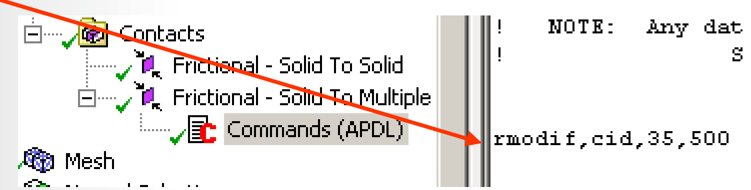


Contact between weld and plates



Contact between the plates

For this contact we further defined a critical Bonding temperature of 500 C using commands. As soon as the temperature at the contact surface (T_c) for closed contact exceeds this bonding temperature, the contact will change to "bonded." The contact status will remain bonded for the rest of analysis, even if the temperature subsequently decreases below the critical value.



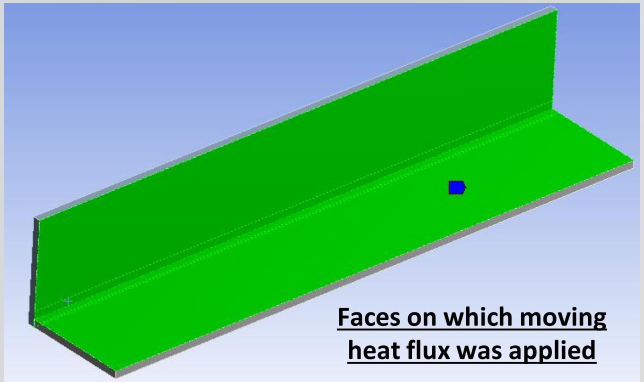
Transient Thermal Analysis

Transient Thermal Analysis

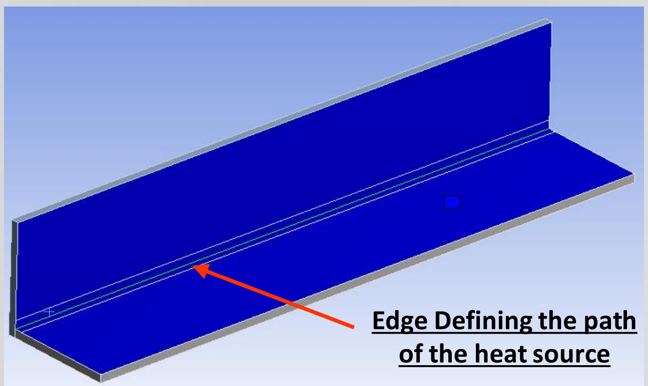
To simulate the thermal field produced by the welding process, it is necessary to model the heat source accurately. In this case, moving heat was modeled using the "Moving_Heat_Flux_R150_v3" ACT extension which is available for download from ANSYS support website.

Please refer to the "Moving_Heat_Flux_R150_v3" ACT extension documentation on how to use this extension.

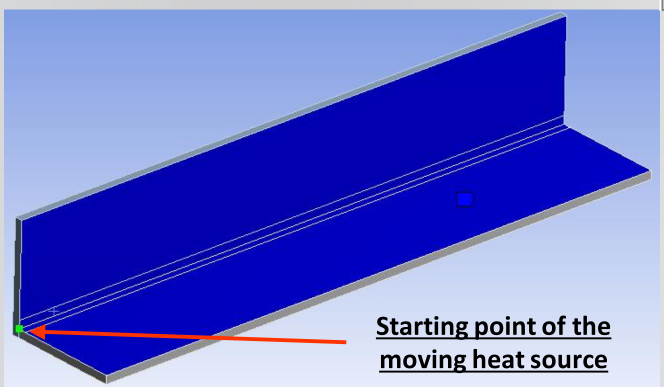
Loading conditions – Moving Heat Flux



Faces on which moving heat flux was applied



Edge Defining the path of the heat source



Starting point of the moving heat source

Moving Heat Flux

etails of "Moving Heat Flux"

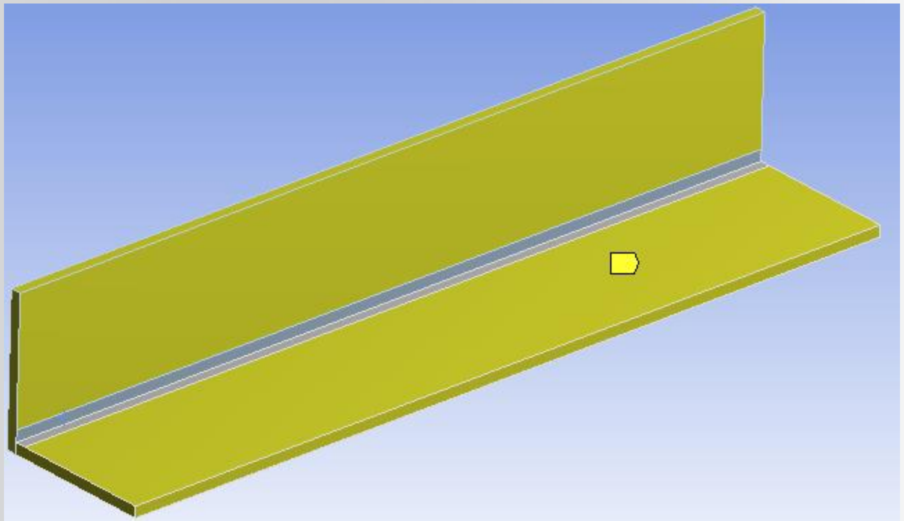
Geometry	
Scoping Method	Geometry Selection
Geometry	4 Faces
Path	
Scoping Method	Geometry Selection
Geometry	1 Edge
Start Point	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Index	1
First Patch?	Yes
Last Patch?	Yes
Velocity	5 [mm sec^-1]
Gassian Heat flux Constant 1	5 [mm]
Gassian Heat flux Constant 2	7.5 [W mm^-1 mm^-1]
Start Time	0 [sec]
End Time	44 [sec]
Number of Segments	100
Minimum Steps for Cooling Phase	20

Velocity of source = 5mm/sec

Start time = 0sec
End Time of the heat source = 44sec

Intensity of laser = 7.5 W/mm^2
Radius of laser beam = 5mm

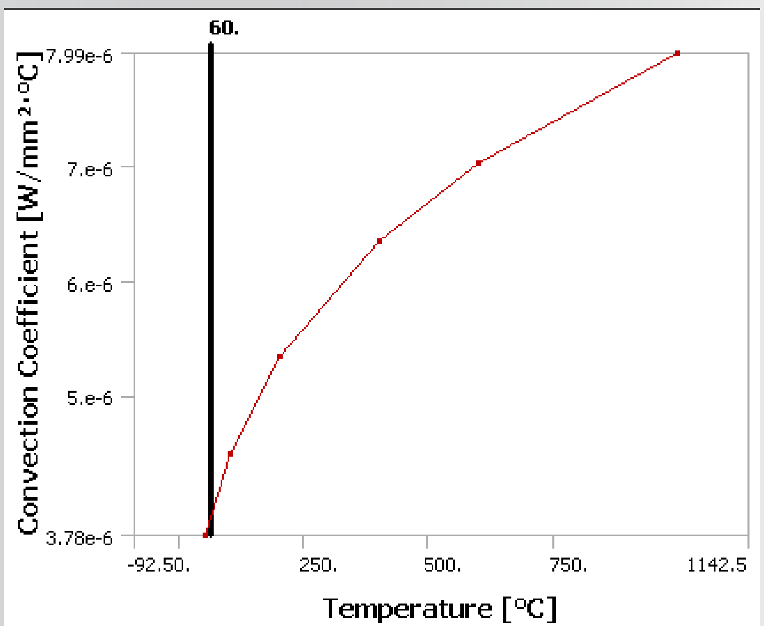
Loading conditions – Convection



Convection

Details of "Convection"

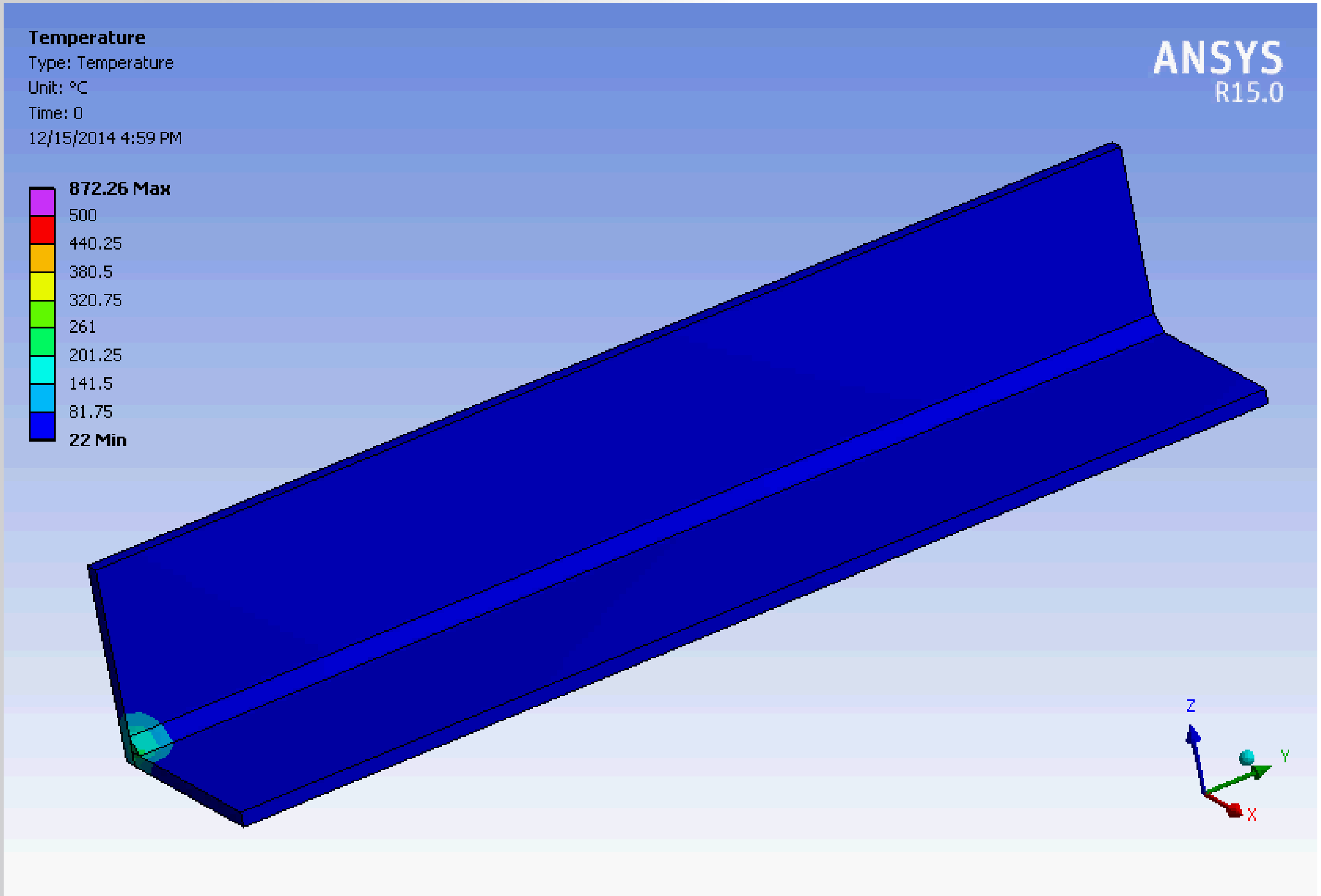
Scope	
Scoping Method	Geometry Selection
Geometry	11 Faces
Definition	
ID (Beta)	127
Type	Convection
Film Coefficient	Tabular Data
Coefficient Type	Average Film Temperature
Ambient Temperature	22. °C (step applied)
Convection Matrix	Program Controlled
Suppressed	No
Fluid Flow (Beta)	No
Edit Data For	Film Coefficient
Tabular Data	
Independent Variable	Temperature
Graph Controls	
X-Axis	Temperature



Tabular Data

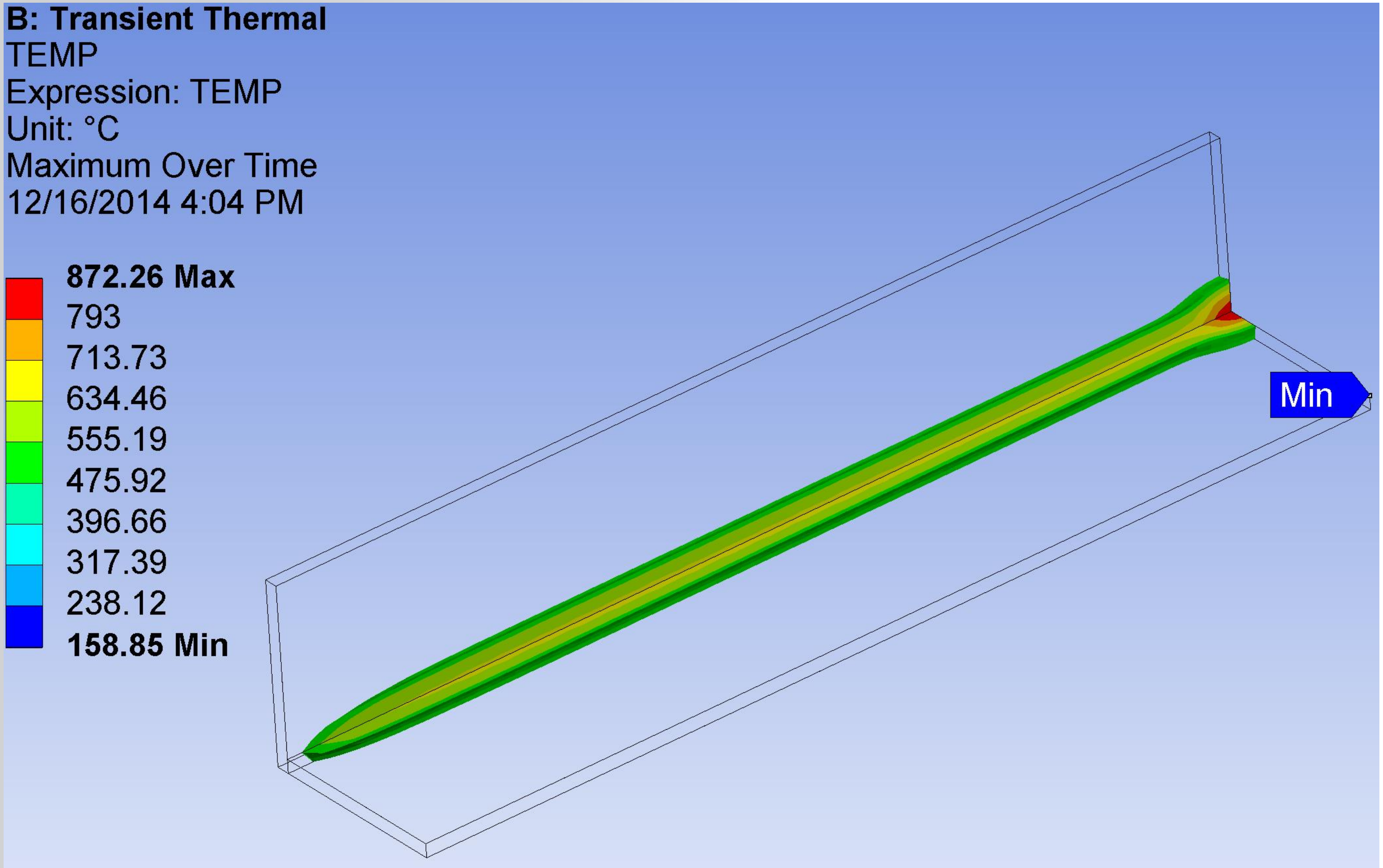
	Temperature [°C]	Convection Coefficient [W/mm²·°C]
1	50.	3.78e-006
2	100.	4.49e-006
3	200.	5.34e-006
4	400.	6.35e-006
5	600.	7.03e-006
6	1000.	7.99e-006
*		

Temperature Field



Heat affected Zone

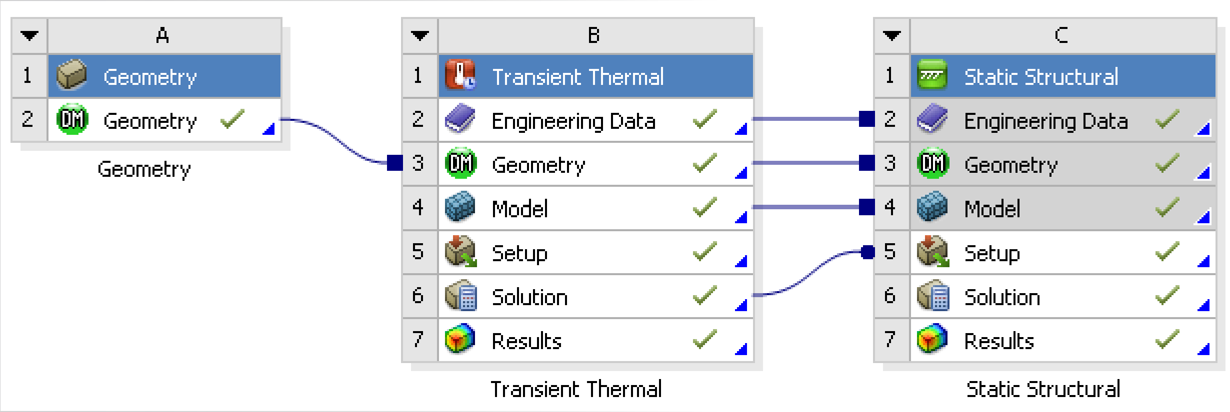
Heat affected is calculated based on the region which has seen 500^{O}C or more temperature during the process.



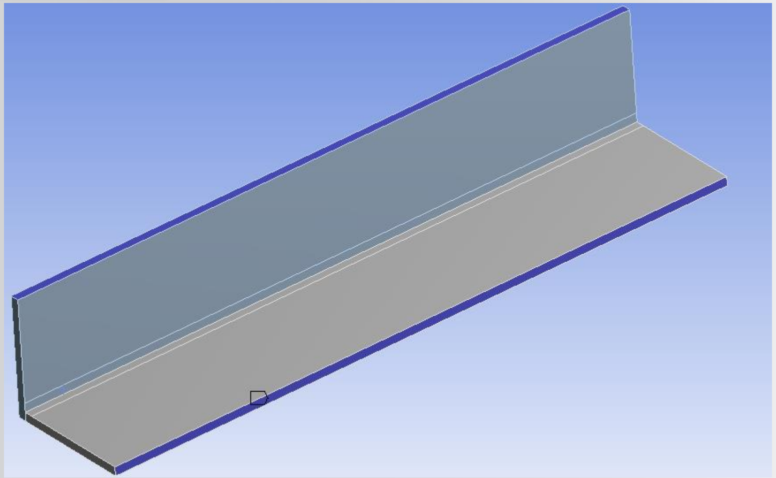
Thermal Stress Analysis

Thermal Stress Analysis

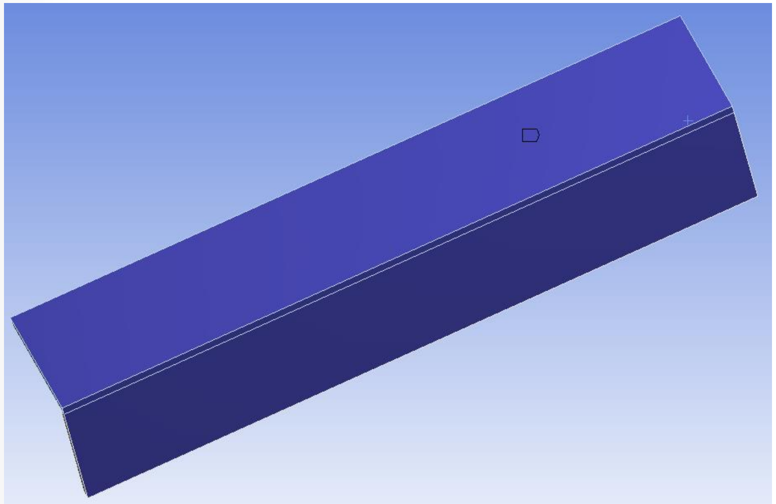
To predict stresses due to the thermal field generated by the transient thermal analysis, drag and drop a "Static Structural" module on "Transient Thermal" as shown below.



Boundary Conditions

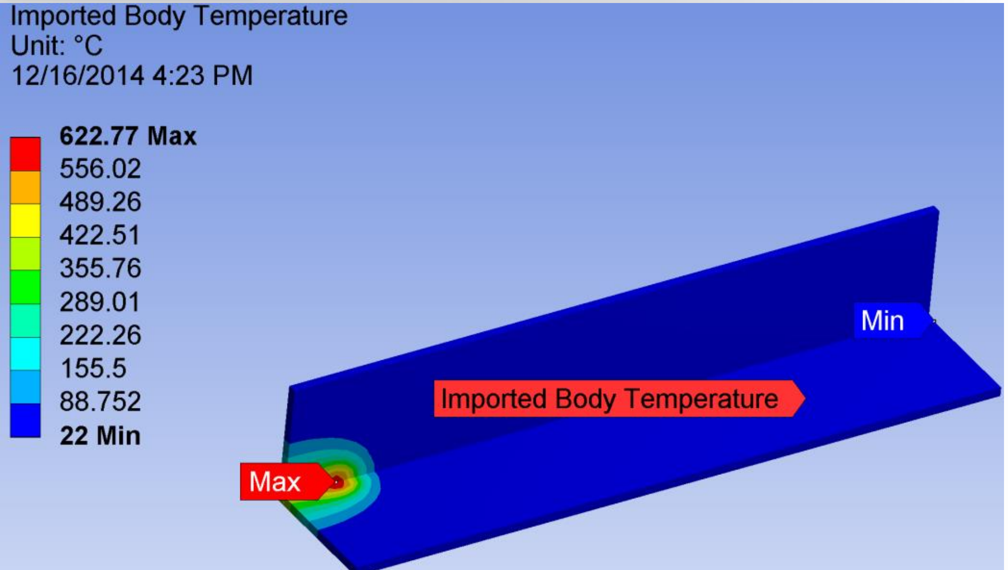


Frictionless Support



Compression Only Support

Loading conditions – Imported Temperature



Temperature from Transient thermal is then imported in the Static Structural for all the steps at which Transient thermal has written data. In total for this case there were 122 steps. Thus we defined 122 steps in Static Structural and imported the data at each step.

Imported Load (B6)
Imported Body Temperature

Details of "Imported Body Temperature"

Scope	
Scoping Method	Geometry Selection
Geometry	3 Bodies
Definition	
Type	Imported Body Temperature
Tabular Loading	Ramped
Suppressed	No
Source Environment	Transient Thermal (B5)
Source Time	Worksheet
Graphics Controls	
By	Active Row
Active Row	10
Beta Options (Beta)	
Show Body Wireframe (Beta)	No

Geometry / Print Preview / Report Preview

Data View

Imported Body Tempera

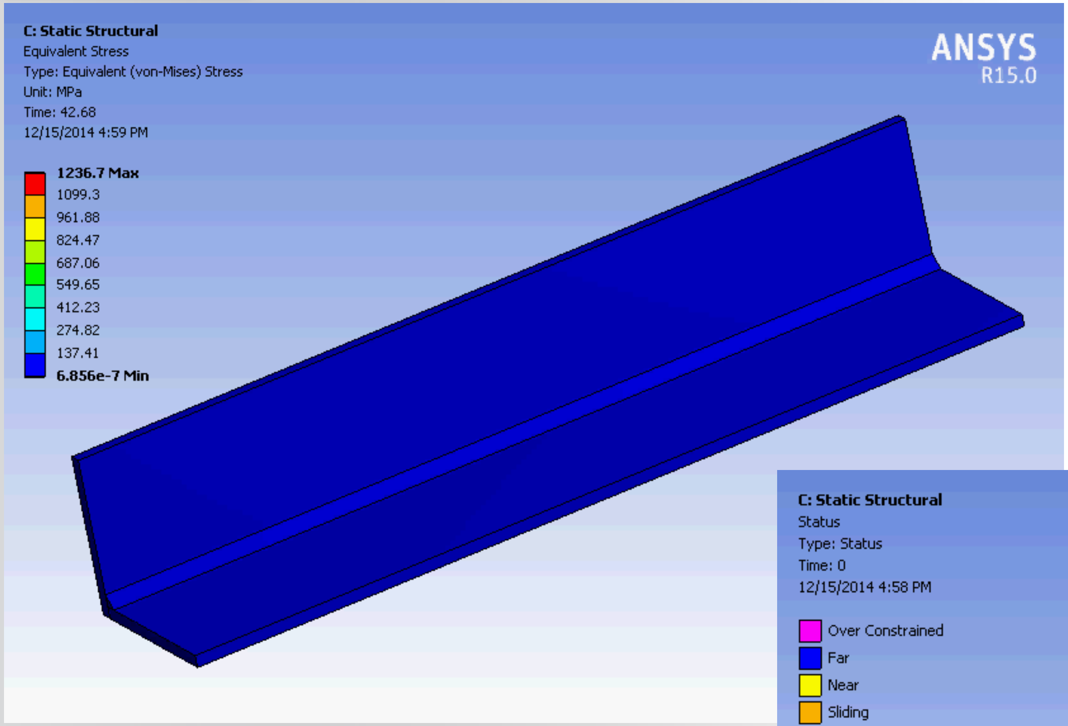
	Source Time (s)	Analysis Time (s)
1	0.44	0.44
2	0.88	0.88
3	1.32	1.32
4	1.76	1.76
5	2.2	2.2
6	2.64	2.64
7	3.08	3.08
8	3.52	3.52
9	3.96	3.96
10	4.4	4.4
11	4.84	4.84
12	5.28	5.28
13	5.72	5.72
14	6.16	6.16
15	6.6	6.6
16	7.04	7.04

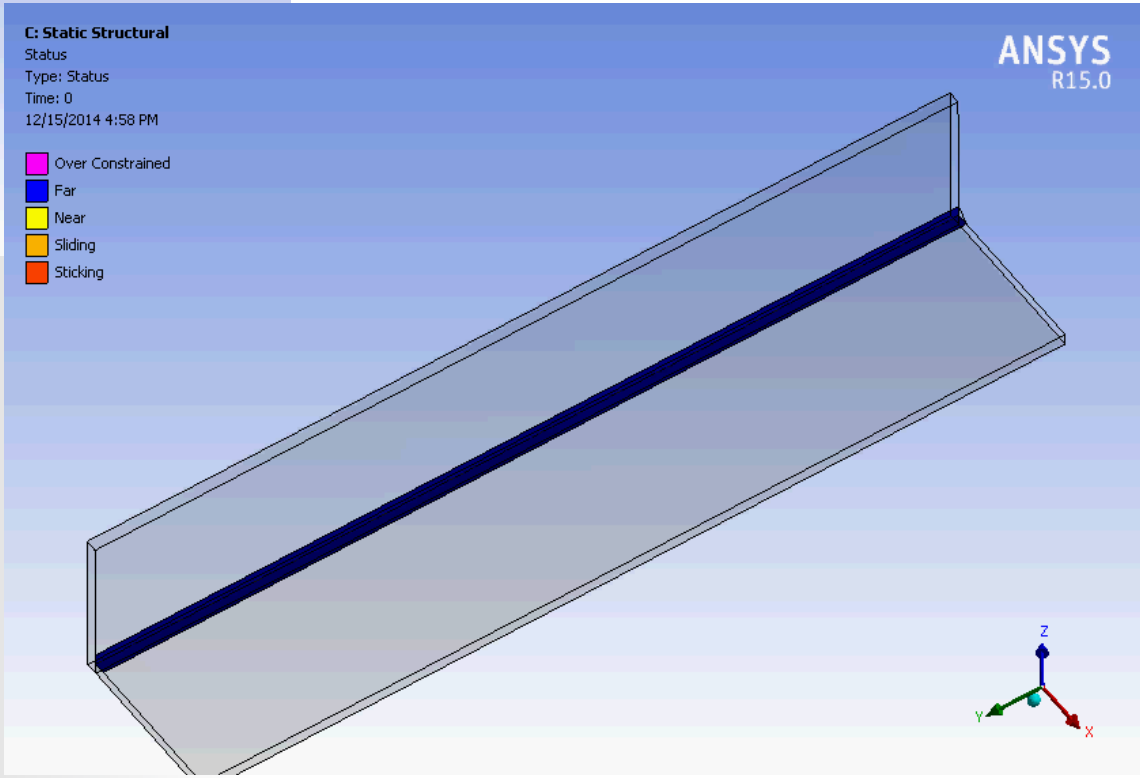
Result Sets from Thermal Analysis

Tabular Data

	Time [s]	Minimum [°C]	Maximum [°C]
1	0.44	22.	242.0284424
2	0.88	22.	375.5061951
3	1.32	22.	457.8466492
4	1.76	22.	501.5859375
5	2.2	22.	542.2215576
6	2.64	21.99999809	570.1467896
7	3.08	21.99999809	584.9587402
8	3.52	21.99999809	593.0780029
9	3.96	21.99999809	612.354187
10	4.4	21.99999809	622.7675171
11	4.84	21.99999809	623.3441772
12	5.28	21.99999809	627.8272095
13	5.72	21.99999809	638.8468628
14	6.16	21.99999809	642.4401855
15	6.6	21.99999809	638.4941406

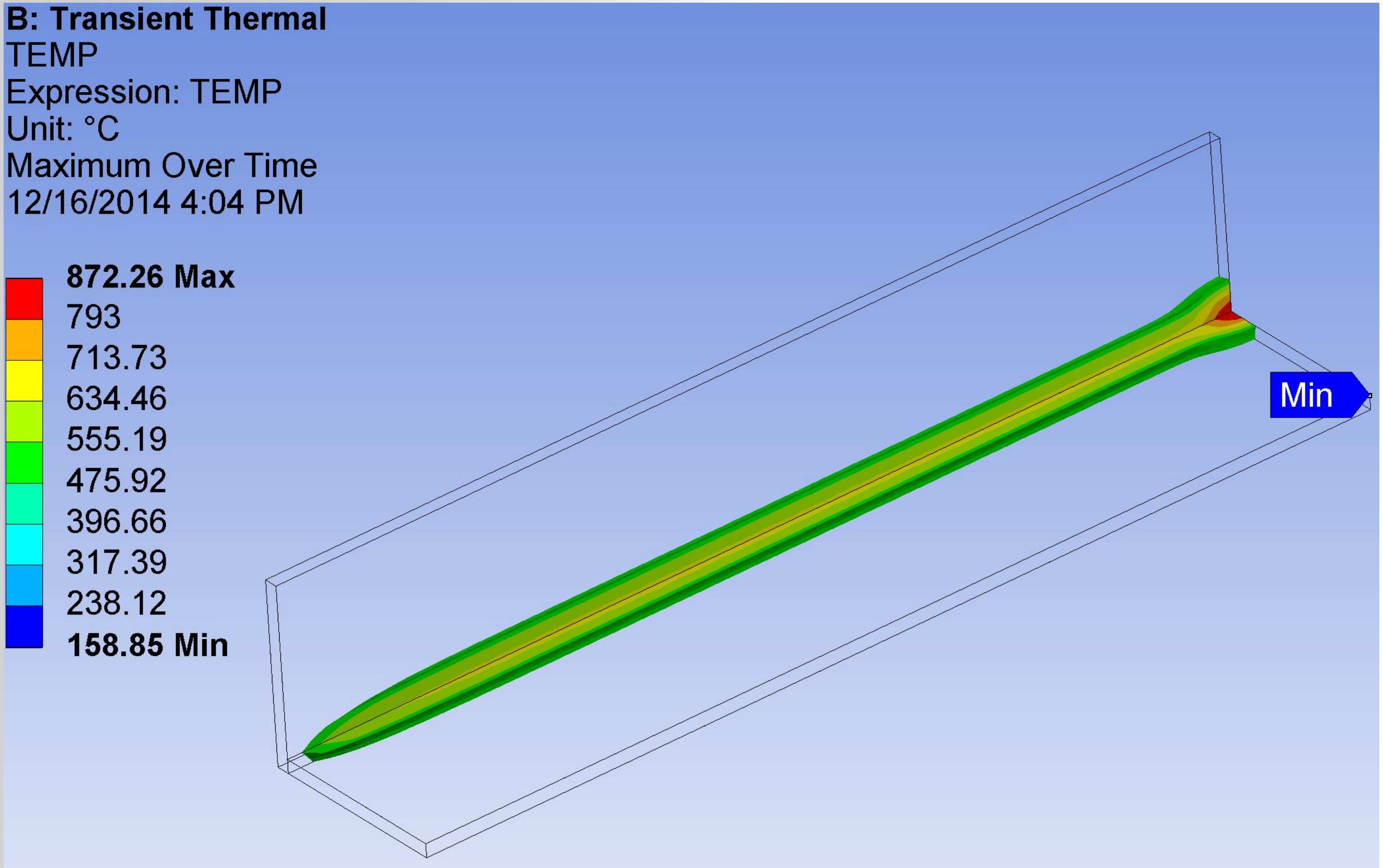
Contact and Stress Results





Heat affected Zone

Heat affected is calculated based on the region which has seen 500^{O}C or more temperature during the process.



Thank You