

Simulation of Assem1

Date: 2026/2/21
Designer: Xinshuo Ze
Study name: Static 1
Analysis type: Static

Description

The Assem1 model represents a reinforced saddle support configuration designed for a horizontal heat exchanger with an operational weight of 1200 kg.

Compared to the lightweight alternative design (Assem2), this configuration utilizes 4 mm thick vertical side support plates to enhance structural stiffness and load-bearing capacity.

The support system consists of a semi-cylindrical cradle providing 180-degree contact with the cylindrical vessel to ensure uniform load distribution across the saddle surface.

Vertical side support plates and a base plate were welded to form a rigid frame structure intended to support the vessel under gravitational loading conditions.

Finite element analysis was conducted to evaluate the structural performance of this reinforced support configuration in terms of stress distribution, displacement, and factor of safety.

Table of Contents

Description	1
Assumptions.....	2
Model Information	3
Study Properties.....	4
Units	4
Material Properties	5
Loads and Fixtures	6
Interaction Information	8
Mesh information.....	9
Resultant Forces	11
Reaction Force Validation	11
Study Results	12
Stress Distribution Interpretation.....	16
Engineering Stress Selection.....	16
Structural Comparison Between Assem1 and Assem2.....	16
Conclusion.....	17



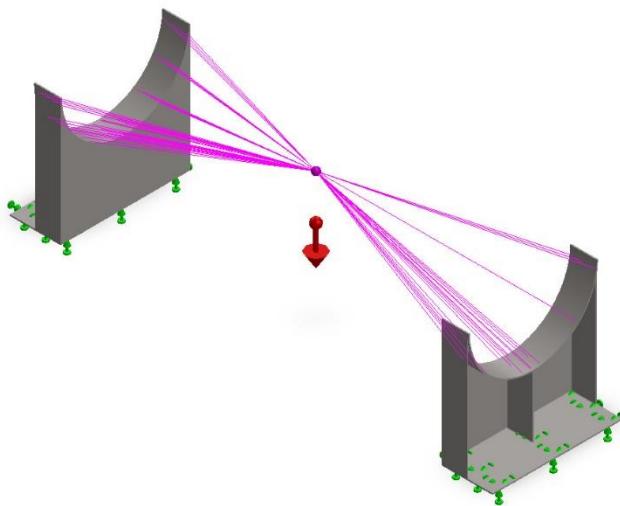
Assumptions

The following assumptions were made in the finite element analysis:

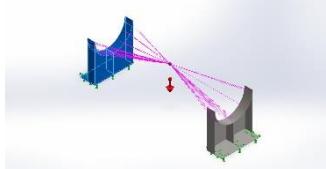
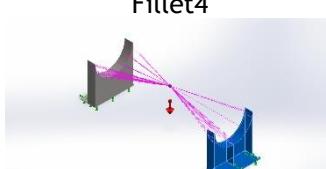
- Linear elastic material behavior was assumed.
- Structural steel was modeled as homogeneous and isotropic.
- Welded connections between structural components were represented using bonded contact.
- Residual stresses due to welding were not considered.
- Manufacturing imperfections and tolerances were neglected.
- Thermal and dynamic effects were not included in this analysis.
- The applied load was assumed to be evenly distributed through a remote mass of 1200 kg.
- Small displacement theory was assumed throughout the analysis.



Model Information



Model name: Assem1
Current Configuration: Default

Solid Bodies			
Document Name and Reference	Treated As	Volumetric Properties	Document Path/Date Modified
Fillet4 	Solid Body	Mass: 16.8774 kg Volume: 0.00216376 m ³ Density: 7,800 kg/m ³ Weight: 165.398 N	D:\SW\GitHub Document\heat-exchanger-support-system\CAD\5 mm-Vertical 3 supports type 1\5 mm-Vertical 3 supports.SLDprt Feb 17 20:06:47 2026
Fillet4 	Solid Body	Mass: 16.8774 kg Volume: 0.00216376 m ³ Density: 7,800 kg/m ³ Weight: 165.398 N	D:\SW\GitHub Document\heat-exchanger-support-system\CAD\5 mm-Vertical 3 supports type 1\5 mm-Vertical 3 supports.SLDprt Feb 17 20:06:47 2026

Study Properties

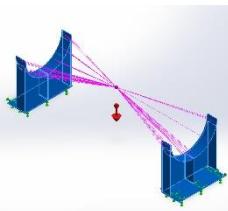
Study name	Static 1
Analysis type	Static
Mesh type	Solid Mesh
Zero strain temperature	298 Kelvin
Include fluid pressure effects from SOLIDWORKS Flow Simulation	Off
Solver type	Automatic
Inplane Effect:	Off
Soft Spring:	Off
Inertial Relief:	Off
Incompatible bonding options	Automatic
Contact penalty stiffness scale factor	1
Large displacement	Off
Compute free body forces	On
Friction	Off
Use Adaptive Method:	Off

Units

Unit system:	SI (MKS)
Length/Displacement	mm
Temperature	Kelvin
Angular velocity	Rad/sec
Pressure/Stress	N/m ²



Material Properties

Model Reference	Properties	Components
	<p>Name: Plain Carbon Steel Model type: Linear Elastic Isotropic Default failure criterion: Max von Mises Stress Yield strength: 2.20594e+08 N/m² Tensile strength: 3.99826e+08 N/m² Elastic modulus: 2.1e+11 N/m² Poisson's ratio: 0.28 Mass density: 7,800 kg/m³ Shear modulus: 7.9e+10 N/m² Thermal expansion coefficient: 1.3e-05 /Kelvin</p>	SolidBody 1(Fillet4)(5 mm-Vertical 3 supports-1), SolidBody 1(Fillet4)(5 mm-Vertical 3 supports-2)
Curve Data:N/A		

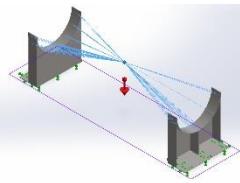


Loads and Fixtures

Fixture name	Fixture Image	Fixture Details															
Fixed-1		Entities: 1 face(s) Type: Fixed Geometry															
Resultant Forces																	
<table border="1"> <thead> <tr> <th>Components</th><th>X</th><th>Y</th><th>Z</th><th>Resultant</th></tr> </thead> <tbody> <tr> <td>Reaction force(N)</td><td>-5.9024e-05</td><td>6,051.66</td><td>-0.000135906</td><td>6,051.66</td></tr> <tr> <td>Reaction Moment(N.m)</td><td>0</td><td>0</td><td>0</td><td>0</td></tr> </tbody> </table>			Components	X	Y	Z	Resultant	Reaction force(N)	-5.9024e-05	6,051.66	-0.000135906	6,051.66	Reaction Moment(N.m)	0	0	0	0
Components	X	Y	Z	Resultant													
Reaction force(N)	-5.9024e-05	6,051.66	-0.000135906	6,051.66													
Reaction Moment(N.m)	0	0	0	0													
Fixed-2		Entities: 1 face(s) Type: Fixed Geometry															
Resultant Forces																	
<table border="1"> <thead> <tr> <th>Components</th><th>X</th><th>Y</th><th>Z</th><th>Resultant</th></tr> </thead> <tbody> <tr> <td>Reaction force(N)</td><td>-6.61789e-05</td><td>6,051.65</td><td>-8.70304e-05</td><td>6,051.65</td></tr> <tr> <td>Reaction Moment(N.m)</td><td>0</td><td>0</td><td>0</td><td>0</td></tr> </tbody> </table>			Components	X	Y	Z	Resultant	Reaction force(N)	-6.61789e-05	6,051.65	-8.70304e-05	6,051.65	Reaction Moment(N.m)	0	0	0	0
Components	X	Y	Z	Resultant													
Reaction force(N)	-6.61789e-05	6,051.65	-8.70304e-05	6,051.65													
Reaction Moment(N.m)	0	0	0	0													

Load name	Load Image	Load Details
Remote Load (Distributed connection)-1		Entities: 6 face(s) Connection: Distributed Type: Weighting Factor: Default (Constant) Coordinate System: Global cartesian coordinates Translational Components: ---, ---, --- Rotational Components: ---, ---, --- Reference coordinates: 750 500 - 255 mm Remote Mass: 1200 kg



		Moment of Inertia: 0,0,0,0,0,0 kg.cm^2
Gravity-1		Reference: Top Plane Values: 0 0 -9.81 Units: m/s^2

Load Representation

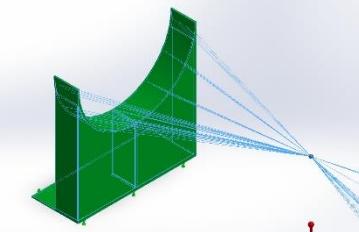
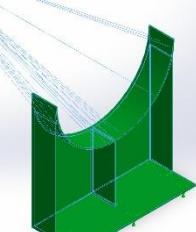
The operational load of the heat exchanger was applied through a distributed remote mass of 1200 kg to simulate the gravitational effect of the supported equipment.

This loading method ensures an even distribution of the operational weight across the saddle support structure, closely representing the real-world support condition of a horizontal vessel.

Gravity was applied in the negative Z-direction to account for the self-weight of the support structure.



Interaction Information

Interaction	Interaction Image	Interaction Properties
Component Interaction-1		Type: Bonded Components: 1 Solid Body (s) Options: Independent mesh
Component Interaction-2		Type: Bonded Components: 1 Solid Body (s) Options: Independent mesh

Welded Connection Modeling

All structural components were connected using bonded contact to represent welded joints in the physical support structure.

This assumption allows load transfer between plates without relative motion, reflecting the rigid behavior of welded steel assemblies.

This modeling approach represents an idealized rigid welded connection and does not account for potential flexibility or residual stress effects in real welded joints.



Mesh information

Mesh type	Solid Mesh
Mesher Used:	Blended curvature-based mesh
Jacobian points for High quality mesh	16 Points
Maximum element size	20 mm
Minimum element size	20 mm
Mesh Quality	High
Remesh failed parts independently	Off
Reuse mesh for identical parts in an assembly (Blended curvature-based mesher only)	Off

Mesh information - Details

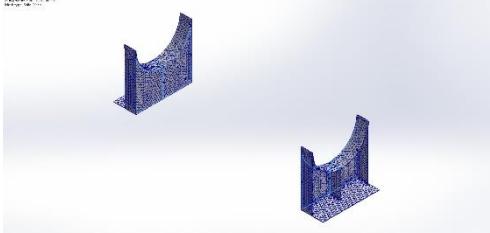
Total Nodes	44932
Total Elements	22014
Maximum Aspect Ratio	24.296
% of elements with Aspect Ratio < 3	26.3
Percentage of elements with Aspect Ratio > 10	0.0727
Percentage of distorted elements	0
Time to complete mesh(hh:mm:ss):	00:00:10
Computer name:	

Although the maximum aspect ratio reached approximately 25-28 in certain regions, the percentage of distorted elements was zero and the majority of elements maintained acceptable geometric quality.

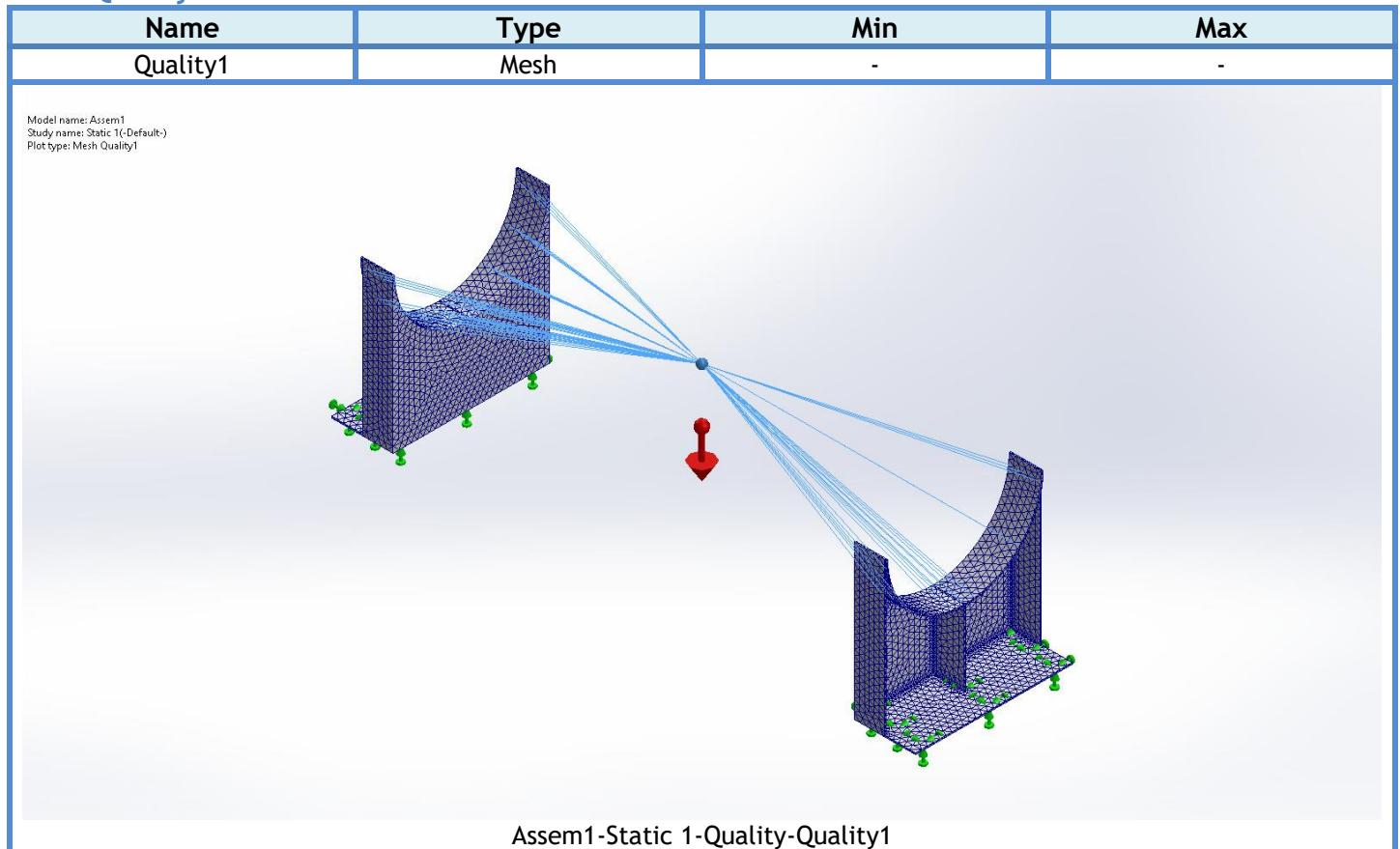
Mesh convergence results further confirmed that the global structural response remained stable despite localized mesh irregularities.

Mesh Control Information:



Mesh Control Name	Mesh Control Image	Mesh Control Details
Control-1		Entities: 25 face(s) Units: mm Size: 10 Ratio: 10

Mesh Quality Plots



Mesh Convergence Study

z	20	15	10	5
Stress (MPa)	24.64	25.3	27.49	28.14



Displacement (mm)	0.149	0.152	0.155	0.157
FOS	8.953	8.72	8.025	7.838

Mesh Quality Assessment

The generated solid mesh exhibited no distorted elements and a low percentage of high aspect ratio elements, ensuring numerical stability of the finite element solution.

Resultant Forces

Reaction forces

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	-9.66964e-05	12,103.3	-0.000239618	12,103.3

Reaction Moments

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N.m	0	0	0	0

Free body forces

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	-0.00014291	216.326	-4.96546e-05	216.326

Free body moments

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N.m	0	0	0	1e-33

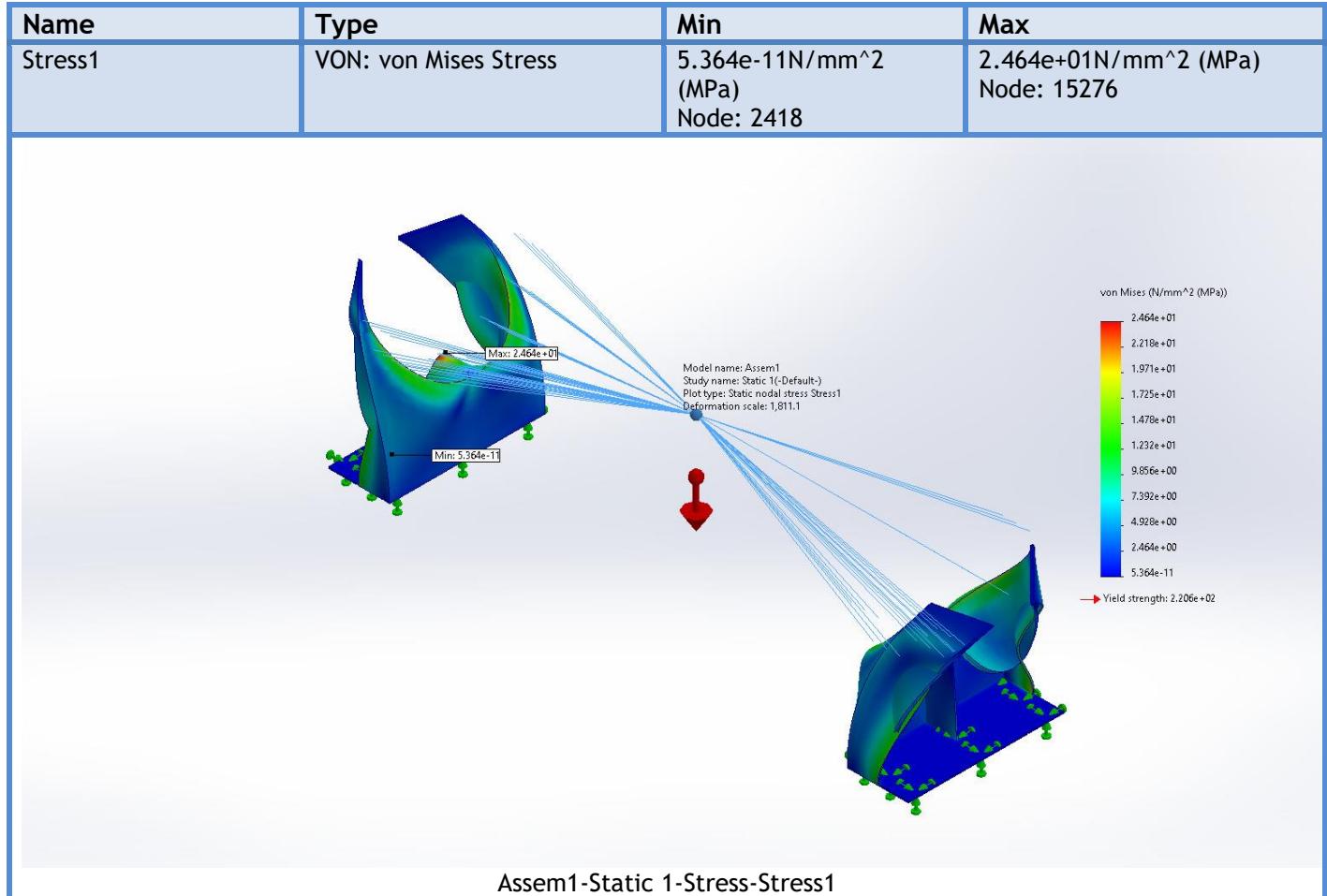
Reaction Force Validation

The total reaction force obtained from the support fixtures was approximately 12,100 N, which closely matches the applied operational load of 1200 kg under gravitational acceleration ($1200 \times 9.81 \approx 11,772$ N).

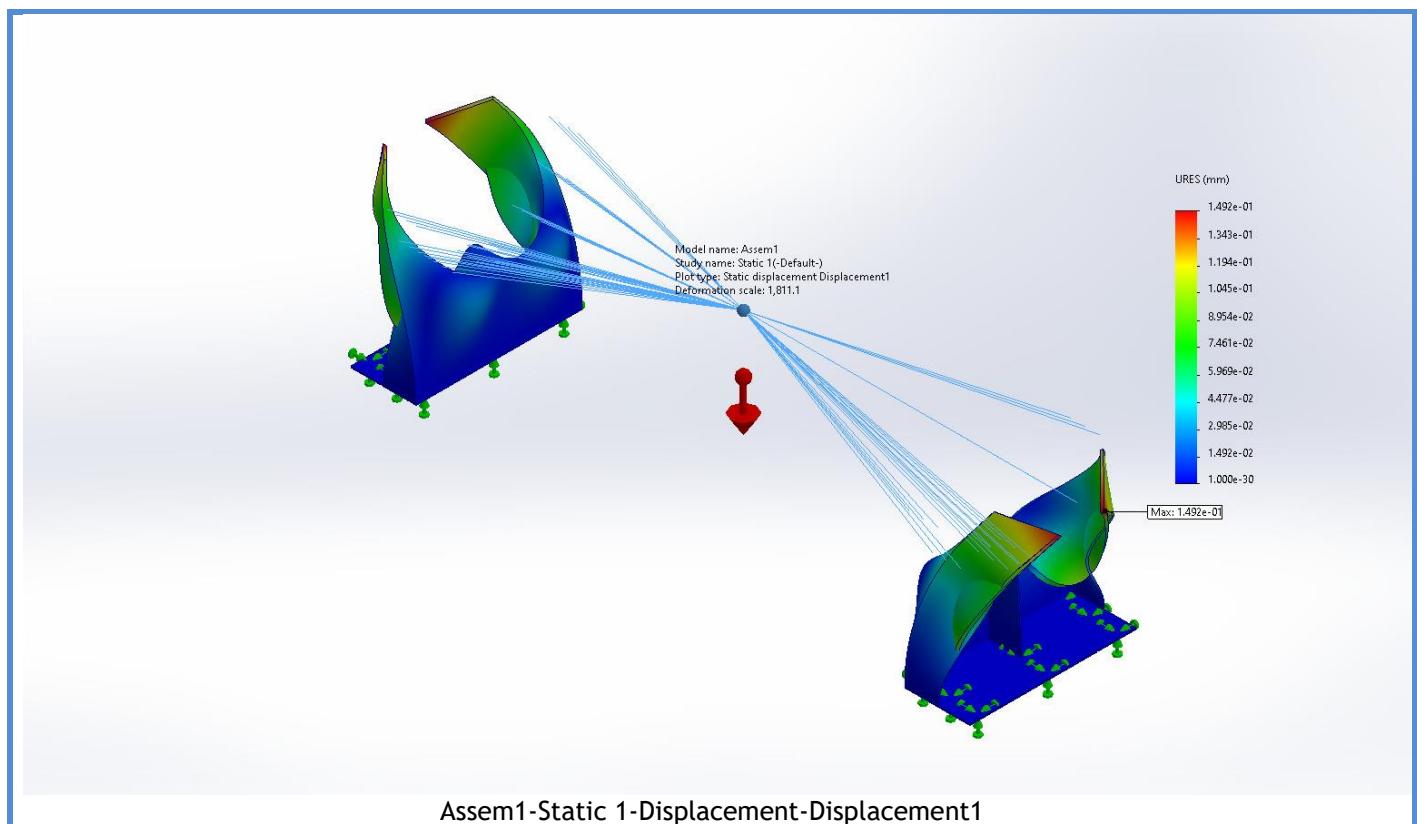
This confirms that the boundary conditions and load transfer mechanisms in the simulation were correctly defined.



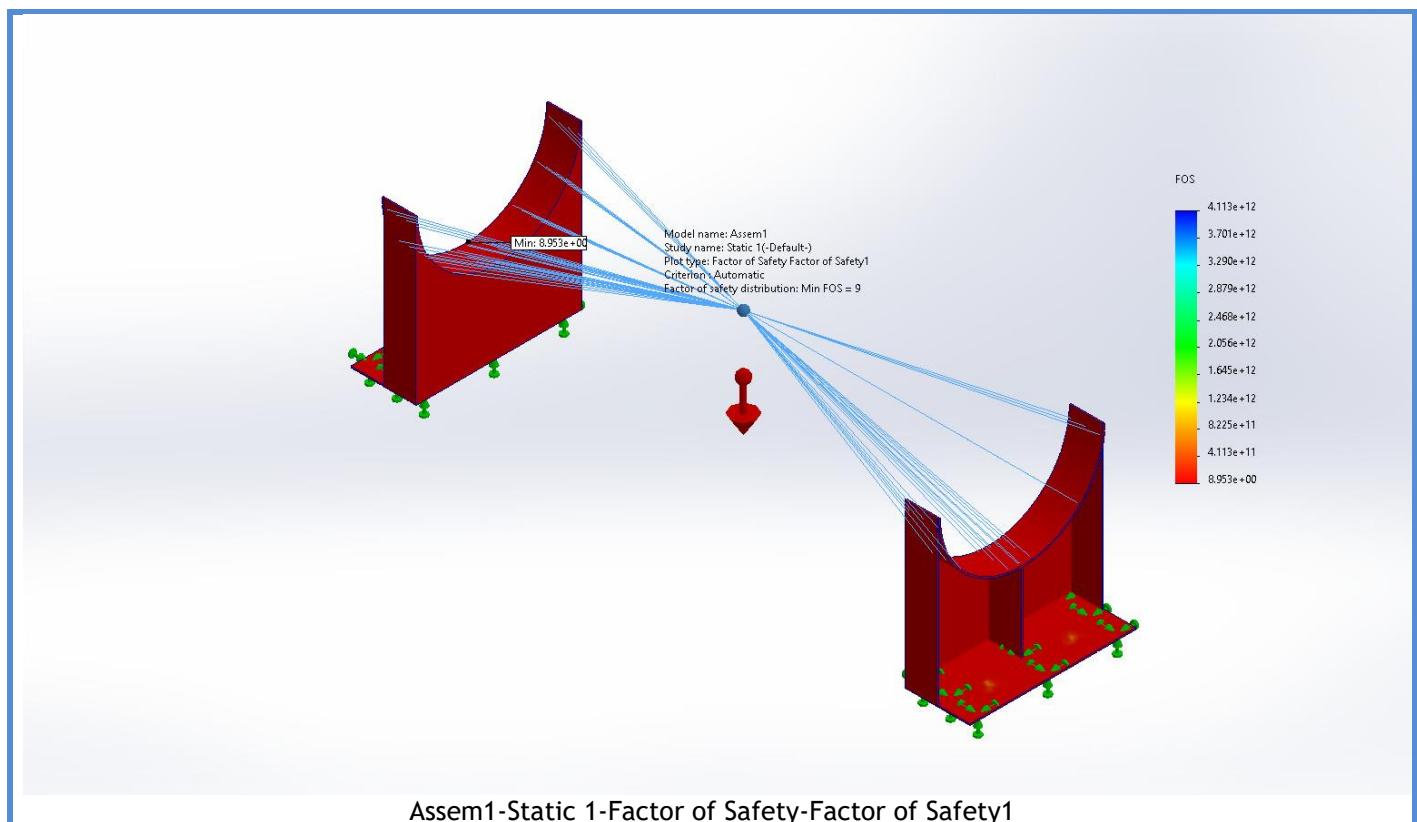
Study Results



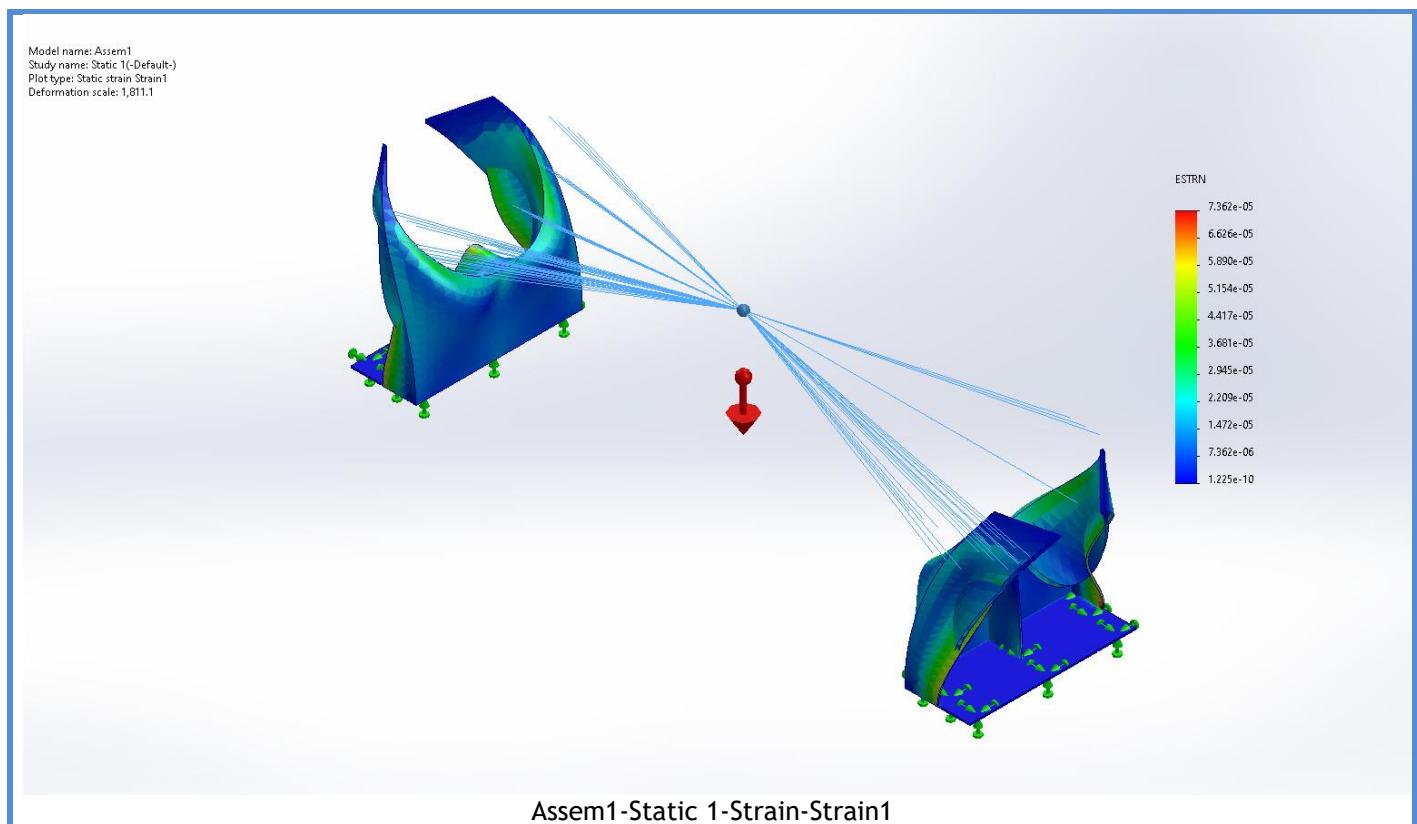
Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0.000e+00mm Node: 27	1.492e-01mm Node: 22469



Name	Type	Min	Max
Factor of Safety1	Automatic	8.953e+00 Node: 15276	4.113e+12 Node: 2418



Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	1.225e-10 Element: 154	7.362e-05 Element: 18979



Mesh Convergence Study

A mesh refinement study was conducted using global mesh sizes of 20 mm, 15 mm, 10 mm, and 5 mm.

The maximum von Mises stress gradually increased with mesh refinement and stabilized between the 10 mm and 5 mm mesh sizes (27.49 MPa and 28.14 MPa respectively).

The variation between these mesh sizes is less than 3%, indicating convergence of the finite element solution.

The structural displacement also showed minimal variation across mesh refinements, confirming convergence of the global stiffness response.

Therefore, the 10 mm global mesh size was selected as an optimal balance between computational efficiency and solution accuracy.



Stress Distribution Interpretation

The stress distribution remained smooth across the saddle support structure as the mesh was refined.

No localized stress concentrations or singularities were observed near welded geometric intersections.

This indicates that the stress response is governed by global structural behavior rather than geometric discontinuity.

Engineering Stress Selection

Based on the mesh convergence study, the representative maximum stress used for structural evaluation corresponds to the converged mesh solution at the 10 mm global mesh size.

Structural Comparison Between Assem1 and Assem2

A structural comparison was conducted between two saddle support configurations with different side support plate thicknesses.

The Assem1 configuration utilized 4 mm thick vertical side support plates, while the Assem2 configuration employed 2 mm thick plates in order to reduce material usage.

Finite element analysis results indicate that increasing the side support plate thickness from 2 mm to 4 mm significantly improved structural performance. The maximum von Mises stress was reduced from approximately 43-44 MPa in Assem2 to below 28 MPa in Assem1.

Similarly, the maximum displacement decreased from approximately 0.22 mm to 0.15 mm.

Although the structural mass increased by approximately 1.34 kg, the minimum factor of safety improved from about 4 to above 7.

This demonstrates a trade-off between structural stiffness and material efficiency, where Assem1 provides enhanced rigidity and safety margin, while Assem2 offers a lighter design with acceptable structural performance.



Conclusion

The finite element analysis results indicate that the Assem1 saddle support structure with 4 mm side support plates provides enhanced strength and stiffness under the applied operational load of 1200 kg.

Mesh convergence was achieved at a global mesh size of 10 mm, ensuring numerical accuracy of the simulation results.

The representative maximum stress remains significantly below the material yield strength, resulting in a minimum factor of safety greater than 7.

Therefore, the proposed support structure offers a high safety margin and improved rigidity for practical application.

