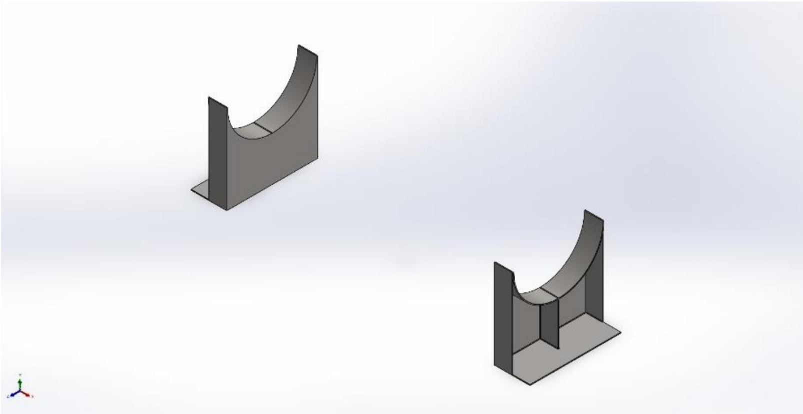


Simulation of Assem2

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

Table of Contents

Description

The Assem2 model represents a lightweight saddle support configuration designed for a horizontal heat exchanger with an operational weight of 1200 kg.

Compared to the reinforced configuration (Assem1), this design employs 2 mm thick vertical side support plates in order to reduce material usage while maintaining acceptable structural performance.

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

Vertical side support plates and a base plate were welded to form a rigid structural frame. Finite element analysis was conducted to evaluate the structural performance of this lightweight support configuration under gravitational loading conditions.

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.

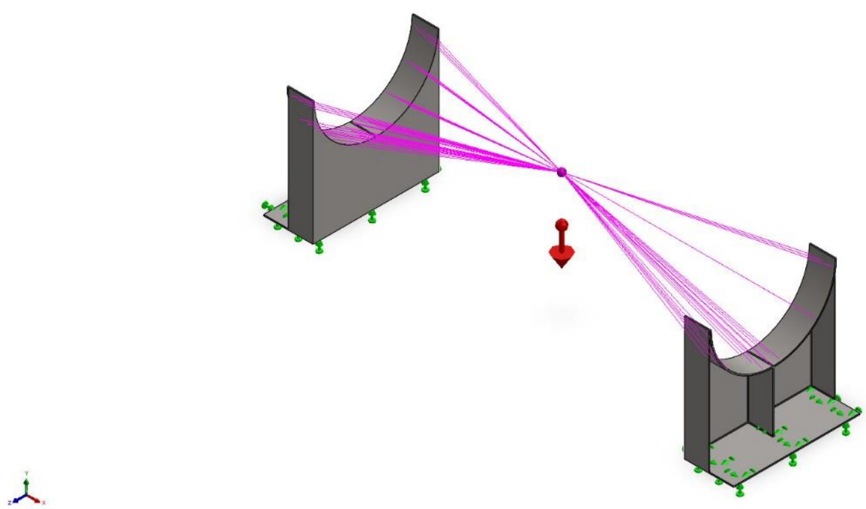
• Manufacturing imperfections and tolerances were neglected.

• Thermal and dynamic effects were not included in this analysis.

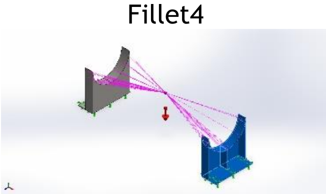
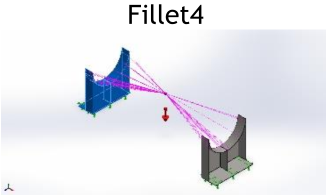
• The applied load was assumed to be evenly distributed through a remote mass of 1200 kg.

• Small displacement theory was assumed throughout the analysis.

Model Information



Model name: Assem2
Current Configuration: Default

Solid Bodies			
Document Name and Reference	**Treated As**	**Volumetric Properties**	**Document Path/Date Modified**
Fillet4 	Solid Body	**Mass:15.5401 kg Volume:0.00199232 m^3 Density:7,800 kg/m^3 Weight:152.293 N**	**D:\SW\GitHub Document\heat-exchanger-support-system\CAD\5 mm-Vertical 3 supports type 2\5 mm-Vertical 3 supports-1.SLDPRT** Feb 18 19:24:59 2026
Fillet4 	Solid Body	**Mass:15.5401 kg Volume:0.00199232 m^3 Density:7,800 kg/m^3 Weight:152.293 N**	**D:\SW\GitHub Document\heat-exchanger-support-system\CAD\5 mm-Vertical 3 supports type 2\5 mm-Vertical 3 supports-1.SLDPRT** Feb 18 19:24:59 2026

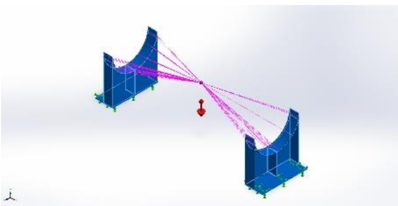
Study Properties

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

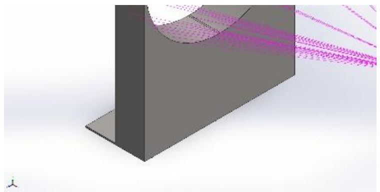
Units

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

Material Properties

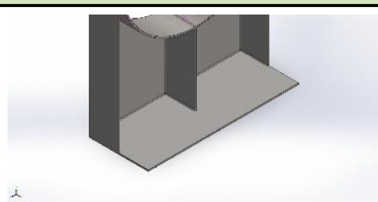
Model Reference	Properties	Components
	**Name:** Plain Carbon Steel **Model type:** Linear Elastic Isotropic **Default failure criterion:** Max von Mises Stress **Yield strength:** 2.20594e+08 N/m^2 **Tensile strength:** 3.99826e+08 N/m^2 **Elastic modulus:** 2.1e+11 N/m^2 **Poisson's ratio:** 0.28 **Mass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

Loads and Fixtures

Fixture name	Fixture Image	Fixture Details
Fixed-3		**Entities:** 1 face(s) **Type:** Fixed Geometry

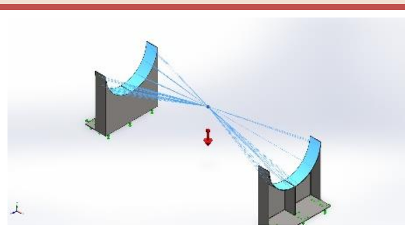
Resultant Forces

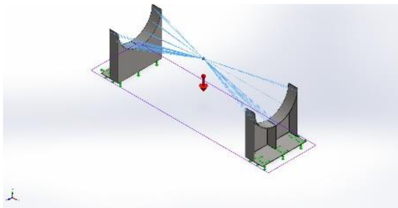
Components	X	Y	Z	Resultant
Reaction force(N)	-6.79654e-05	6,038.54	-9.01854e-05	6,038.54
Reaction Moment(N.m)	0	0	0	0

Fixture name	Fixture Image	Fixture Details
Fixed-4		**Entities:** 1 face(s) **Type:** Fixed Geometry

Resultant Forces

Components	X	Y	Z	Resultant
Reaction force(N)	-0.000226027	6,038.53	3.27407e-05	6,038.53
Reaction Moment(N.m)	0	0	0	0

Load name	Load Image	Load Details
Remote Load (Distributed connection)-1		**Entities:** 6 face(s) **Connection Type:** Distributed **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

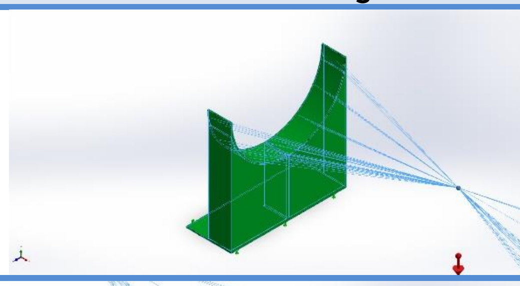
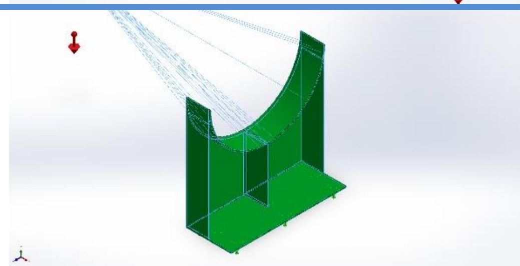
Load Representation

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

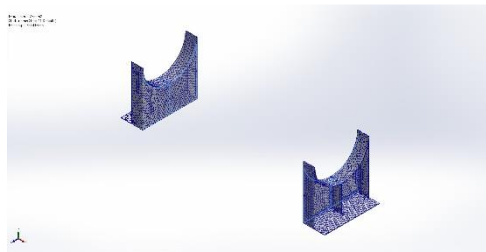
Mesh information - Details

Total Nodes	45157
Total Elements	22112
Maximum Aspect Ratio	27.993
% of elements with Aspect Ratio < 3	25.1
Percentage of elements with Aspect Ratio > 10	0.561
Percentage of distorted elements	0
Time to complete mesh(hh;mm;ss):	00:00:06
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.

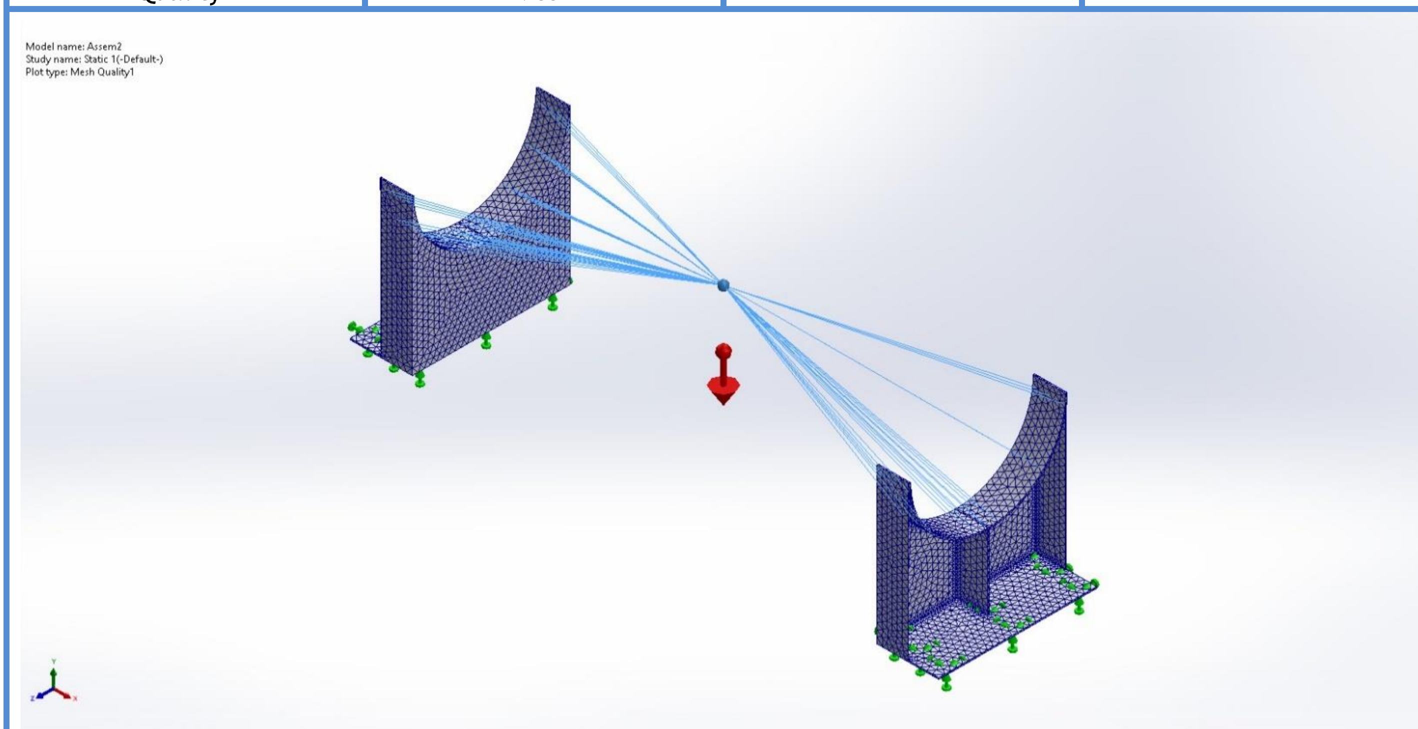
Mesh convergence results further confirmed that the global structural response remained stable despite localized mesh irregularities.

Mesh Control Information:

Mesh Control Name	Mesh Control Image	Mesh Control Details
Control-1		**Entities:** 24 face(s) **Units:** mm **Size:** 10 **Ratio:** 10

Mesh Quality Plots

Name	Type	Min	Max
Quality1	Mesh	-	-



Assem2-Static 1-Quality-Quality1

Mesh Convergence Study

z	20	15	10	5
Stress (MPa)	39.19	43.7	43.73	54.36

Displacement (mm)	0.213	0.218	0.221	0.225
FOS	5.629	5.048	5.044	4.058

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	-0.000281588	12,077	-5.96678e-05	12,077

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	-0.000238673	199.635	-0.000290457	199.635

Free body moments

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

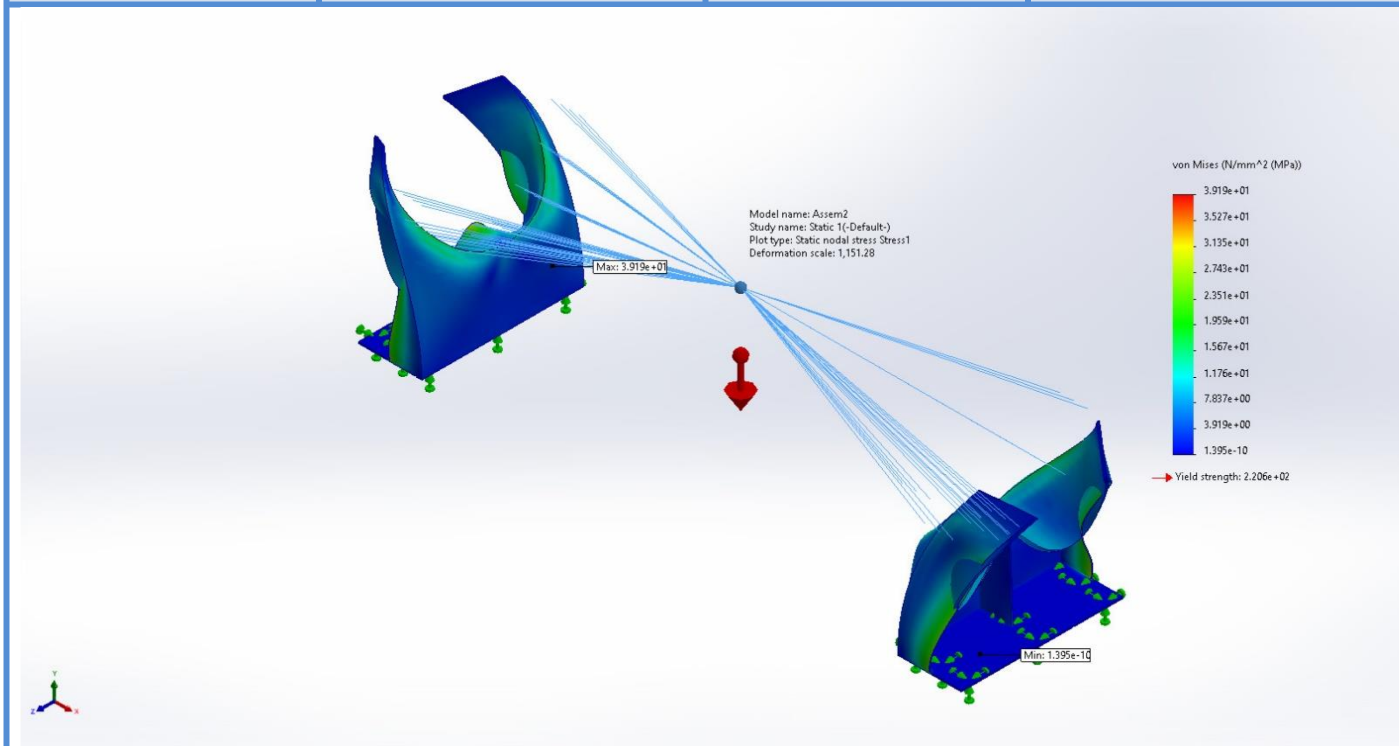
Reaction Force Validation

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.

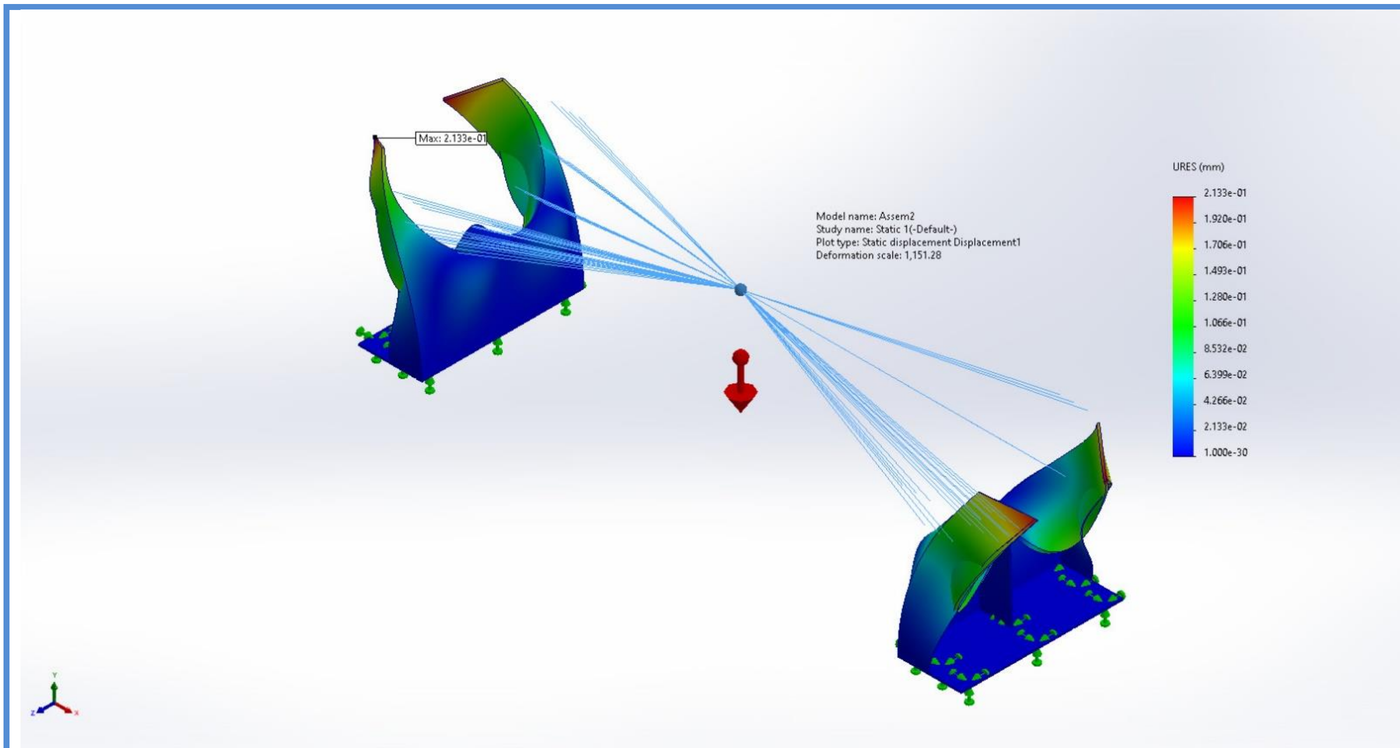
Study Results

Name	Type	Min	Max
Stress1	VON: von Mises Stress	1.395e-10N/mm^2 (MPa) Node: 12656	3.919e+01N/mm^2 (MPa) Node: 31502



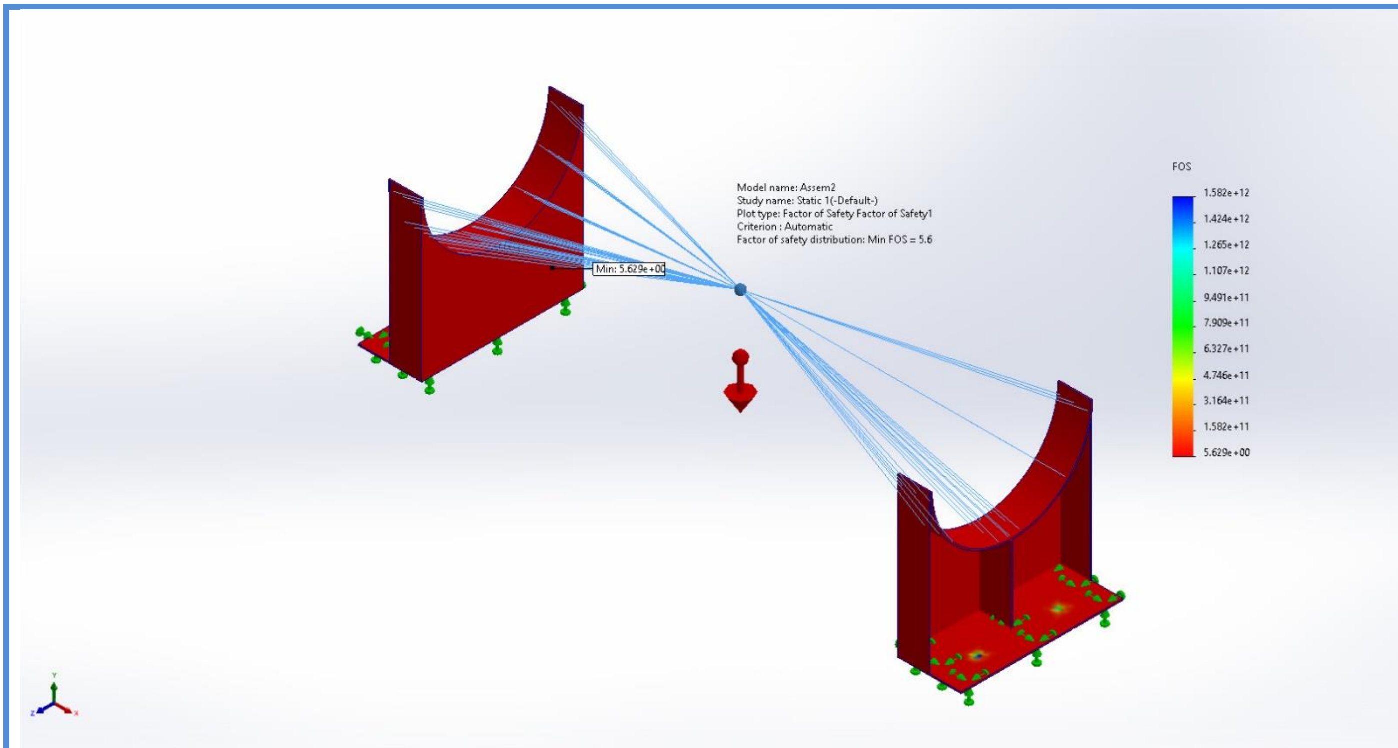
Assem2-Static 1-Stress-Stress1

Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0.000e+00mm Node: 27	2.133e-01mm Node: 22609



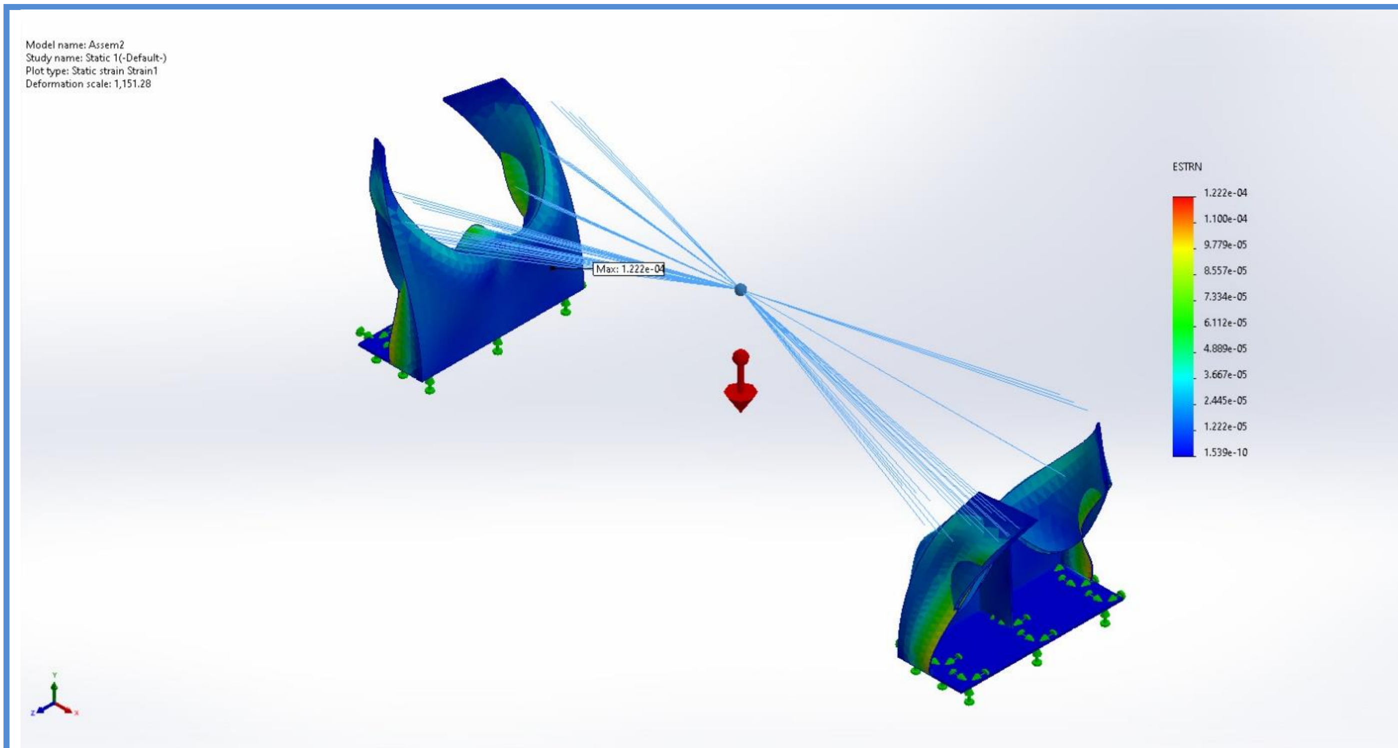
Assem2-Static 1-Displacement-Displacement1

Name	Type	Min	Max
Factor of Safety1	Automatic	5.629e+00 Node: 31502	1.582e+12 Node: 12656



Assem2-Static 1-Factor of Safety-Factor of Safety1

Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	1.539e-10 Element: 6285	1.222e-04 Element: 12248



Assem2-Static 1-Strain-Strain1

Mesh Convergence Study

A mesh refinement study was conducted using global mesh sizes of 20 mm, 15 mm, 10 mm, and 5 mm to assess the numerical accuracy of the finite element results.

The maximum von Mises stress stabilized between the 15 mm and 10 mm mesh sizes (43.70 MPa and 43.73 MPa respectively), indicating convergence of the global structural response. Further refinement to 5 mm resulted in a localized increase in peak stress to 54.36 MPa, which was confined to the sharp edge between the base plate and side support.

This increase is attributed to stress singularity due to geometric discontinuity at welded connections. The overall structural displacement remained stable across mesh refinements (variation < 6%), confirming that the global stiffness response has converged.

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

Stress Distribution Interpretation

The peak stress observed at the 5 mm mesh size was localized at the sharp geometric intersection between the base plate and vertical side support.

This stress concentration is attributed to geometric discontinuity at welded joints and represents a typical stress singularity in finite element modeling.

In practical welded structures, such intersections are manufactured with finite fillet radii rather than perfectly sharp edges.

Therefore, the localized peak stress obtained from the refined mesh does not represent the actual structural stress state.

Engineering Stress Selection

Based on the mesh convergence study, the global structural stress stabilized between the 15 mm and 10 mm mesh sizes.

Therefore, the representative maximum stress adopted for structural evaluation was taken as approximately 43–44 MPa, corresponding to the converged mesh solution at 10 mm global mesh size.

This value was used for subsequent factor of safety assessment.

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 Assem2 saddle support structure with 2 mm side support plates provides acceptable strength and stiffness under the applied operational load of 1200 kg.

Mesh convergence was achieved at a global mesh size of 10 mm, and localized stress concentrations were identified as non-critical stress singularities and do not indicate structural failure under the applied loading conditions.

The representative maximum stress remains significantly below the material yield strength, resulting in a minimum factor of safety greater than 4.

Therefore, the proposed support structure represents a lightweight alternative suitable for applications where material efficiency is prioritized.