IJoCED

Indonesian Journal of Computing, Engineering, and Design

Journal homepage: http://ojs.sampoernauniversity.ac.id/index.php/IJOCED

Parametric Finite Element Method-Based Analysis of Curved Sandwich Structures under Projectile Impact

Galang Bagaskara[1], *Budi Arivfianto*[1], *Muslim Mahardika*[1], *Zebing Mao*[2], *Ardi Wiranata*[1*]

[1] Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Gadjah Mada, Jl. Grafika 2, Yogyakarta 55281, Indonesia

[2] Zijingang Campus, Zhejiang University, Sandun Town, Xihu District, Hangzhou, Zhejiang, China

*Corresponding email: ardi.wiranata@ugm.ac.id

ABSTRACT

This study aims to develop a Python and Gmsh-based program for automating the generation of finite element method (FEM) models and integration with the OpenRadioss solver, enabling more efficient creation of a FEM model for a curved corrugated sandwich structure under projectile impact. Based on the research conducted, the program has been successfully integrated with OpenRadioss and successfully simulated. The simulation results show a graph that closely matches experimental data, indicating that the generated finite element model is acceptable. The time required to generate the model is approximately ±73 s, while conventional methods require at least ±480 s. Additionally, this program has been compared with the latest Graphical User Interfaces (GUI) provided by OpenRadioss, demonstrating the advantages of the developed program in terms of time efficiency, ease of use, and its ability to handle design variations more quickly.

ARTICLE INFO

Article History:
Received 15 May 2024
Revised 11 Jun 2025
Accepted 25 Jun 2025
Available online 21 Oct 2025

Keywords:
Corrugate core,
Curve sandwich,
Dynamic impact,
FEM,
Parametric model.

1. INTRODUCTION

In impact energy absorption applications, sandwich structures are commonly used due to their superior energy absorption capability compared to a monolithic structure (Hashemi et al., 2024), In general, a sandwich structure consists of two main components that influence energy absorption capability including the face sheets and core (Tarlochan, 2021). In the energy absorption of sandwich structure, it is known that the structural core design influences the energy absorption capability, for example thickness face sheet, cell size, and wall thickness (Sun et al., 2021).

In certain applications, the surface of the sandwich structure is not always flat, as commonly used, but curved sandwich structures are also employed (Bozkurt, 2024). When the sandwich structures take a curved form, it presents significant manufacturing challenges in the production of curved sandwich structure (Um et al., 2025). One possible solution is to use additive manufacturing (AM) as it allows for the fabrication of complex geometries and the overcoming of the challenges associated with manufacturing intricate shapes. When using AM, the initial step is to create an object using a 3D computer aided design (CAD) model (Zhou et al., 2024). However, this process can be a challenge due to the complex curved geometries of sandwich structures, which make CAD modelling more difficult (Peterson & Kaur, 2024) and time-consuming. Therefore, a computer program is needed to automatically design a curved sandwich structure with a corrugated core pattern.

The assessment of impact energy absorption in curved sandwich structures are commonly conducted using the finite element method (FEM) (Wang et al., 2024). However, when simulation impact projectile on curved sandwich structure, a major challenge is the time-consuming process of setting up simulations, such as 3D modeling using CAD and meshing, particularly for corrugated structure. Therefore, a computer program is needed to streamline the finite element model generation. The combination of an automatically designed corrugated sandwich structure with an automatically generated finite element model for simulation curved sandwich structure is an effective approach to reduce the time required from CAD design to finite element modeling. However, a major challenge lies in integrating these two programs. Therefore, this research aims to integrate the automatic design of the curved sandwich structure with finite element modelling for projectile impact dynamic case.

2. RESEARCH METHODS

This program is designed to simplify the process for the user in creating a finite element model specifically for projectile impact cases on curved corrugated sandwich structures. When using this program, users do not utilize CAD software to create a 3D model. Instead, they only need to input parameters. The logical scheme of this program can be seen in **Figure 1**. This approach offers significant advantages, as it greatly reduces the time required and is user-friendly, especially for those unfamiliar with CAD software.

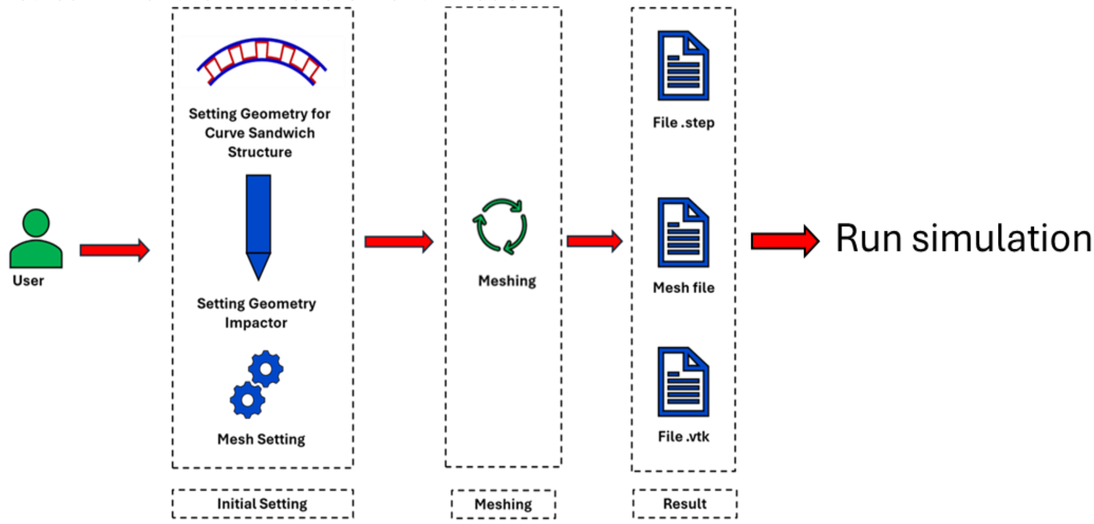

Figure 1. Logical Scheme

In this research, the impact projectile testing model refers to the study conducted by (Yang et al., 2020). The testing model can be seen in **Figure 2**. There are a total of four components, including two face sheets (top sheet and bottom sheet), the core, and the projectile, to create 3D model for simulations. This program requires input parameters during the initial setting stage, including setting the geometry for curved corrugated sandwich structure, setting the geometry of impactor, and mesh setting. The output of this program consists of three file formats: .step, a widely used format for design engineering; .vtk, a format for visualizing mesh results; compatible with software such as ParaView; and mesh file (.msh). In this research, this program is written in Python where the Gmsh mesh generator library is used as a tool for generating geometry and meshing in the 3D model (Geuzaine & Remacle, 2009). Meanwhile, OpenRadioss code is used for the non linear FEM solver.

Creating a 3D model using Gmsh involves several steps. First, points must be defined by specifying their x, y, and z coordinates. Once these points are created, lines can be generated by connecting these points. After the line is defined, a surface can be created by forming a curve loop, which must be a closed boundary without any gaps. Once the curve loop is completed, the surface can then generated. The next step is to create a volume by using the extrude feature on the previously generated surface. **Figure 2a** shows the surface to be extruded along L, while **Figure 2b** illustrates the result after extrusion. Using the extrusion process, The result of 3D modelling is shown in **Figure 2c**. The process of creating the projectile model is similar to that of the previously discussed corrugated sandwich structure. However, the key difference is that to generate the 3D projectile, the surface is revolved instead of extruded.

In this program, the core structure with a corrugated pattern is classified into several types of corrugate sandwich structure based on its cross-sectional such as trapezoidal and rectangular (Bozkurt, 2024) (see **Figure 3a, 3b**) . This program provides three types of projectiles shape such as hemispherical, flat, and conical. The projectiles shape are shown in **Figure 3c** until **Figure 3e**. This program also includes the GUI, which is shown in **Figure 4**, the GUI consist at least four main stage, the first stage involving inputing geometric data, which includes parameters required to generate curved corrugate sandwich structures, such as radius, angle, etc.

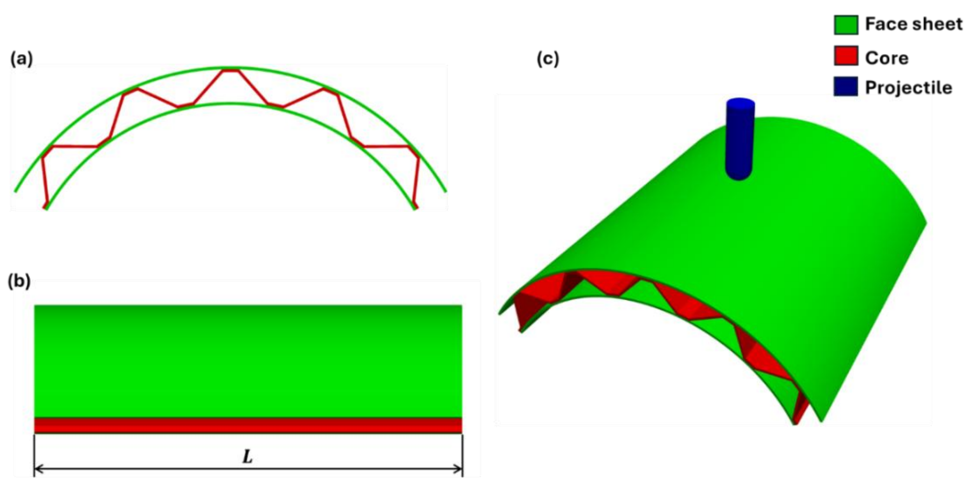

Figure 2. 3D model impact test; (a). Front view, (b). Side view, (c). Isometric view.

The next stage is to input data for generating the projectile model, such as radius of the projectile, length of the projectile and projectile mass. Then, the next stage is mesh configuration, where the user inputs the number of elements to be used. In the final stage, the user selects the folder directory, where the folder contains the output files. Next, to ensure that the dimension of the curved corrugated sandwich structure matches the input parameter, a test case is conducted by generating the structure using the parameter listed in **Table 1**. Then, Autodesk Inventor is used to verify whether the dimensions of the generated curved corrugated sandwich structure are accurate.

Table 1. Example parameter

Parameter	Value	Unit	Description
Radius	71	mm	-
Angle	30	Deg	-
Core Shape	Trapezoidal	-	-
Core Height	11	mm	-
Length	160	mm	-
NUCS	5	-	Number of a unit cell
TUS	1	mm	Thickness Upper sheet
TBS	1	mm	Thickness Bot sheet
TCW	1	mm	Thickness Core wall
Impactor			
Impactor Shape	Hemispherical	-	-
Impactor radius	6	mm	-
Impactor Length	36	mm	-

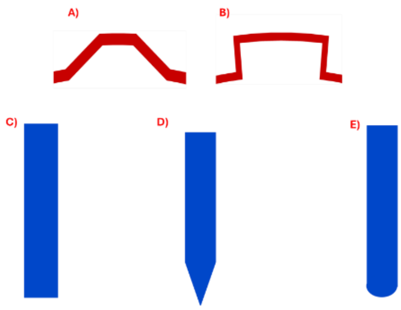

Figure 3. Corrugate core shape, Impactor shape, GUI; (a) Trapezoidal, (b) Rectangular, (c) Flat, (d) Conical,(e) Hemispherical

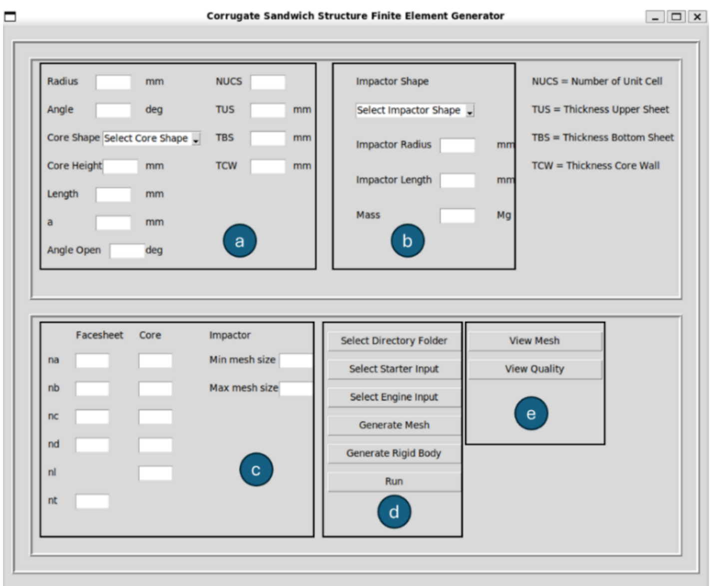

Figure 4. Main GUI; (a). Curved sandwich structure geometry, (b). Impactor geometry, (c). Mesh setting, (d). Directory folder and Running, (e). Visualize mesh result and check quality mesh.

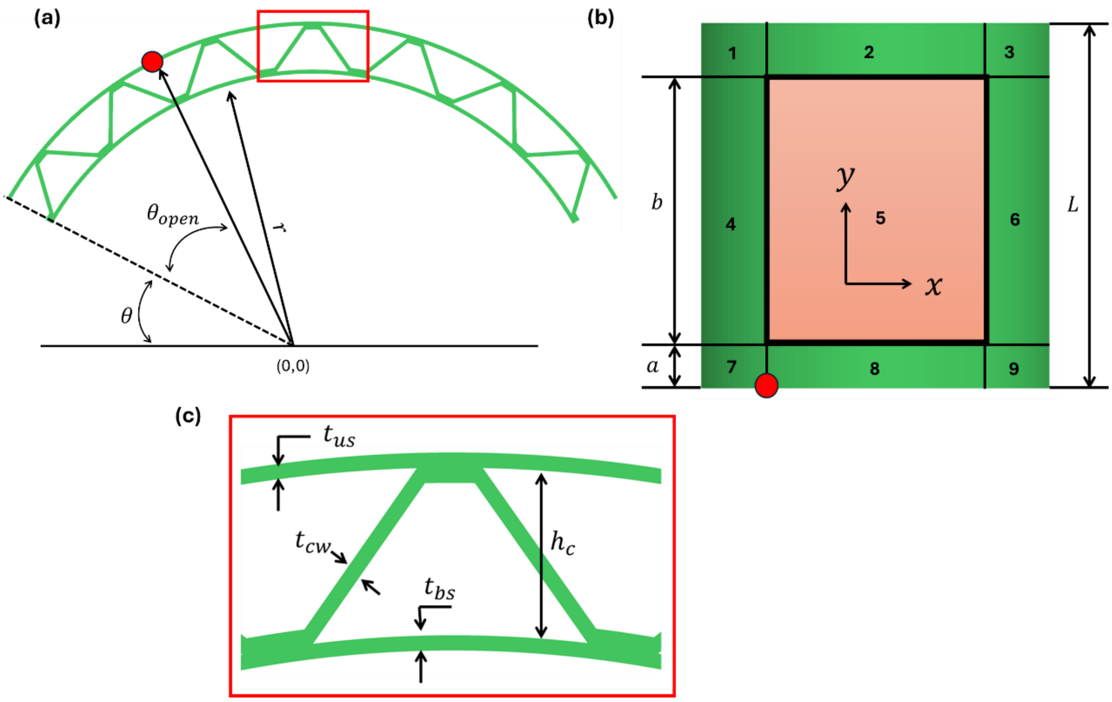

Figure 5. Parameter detail of curved sandwich structure; a. Front view, b. Top view c. Unit cell

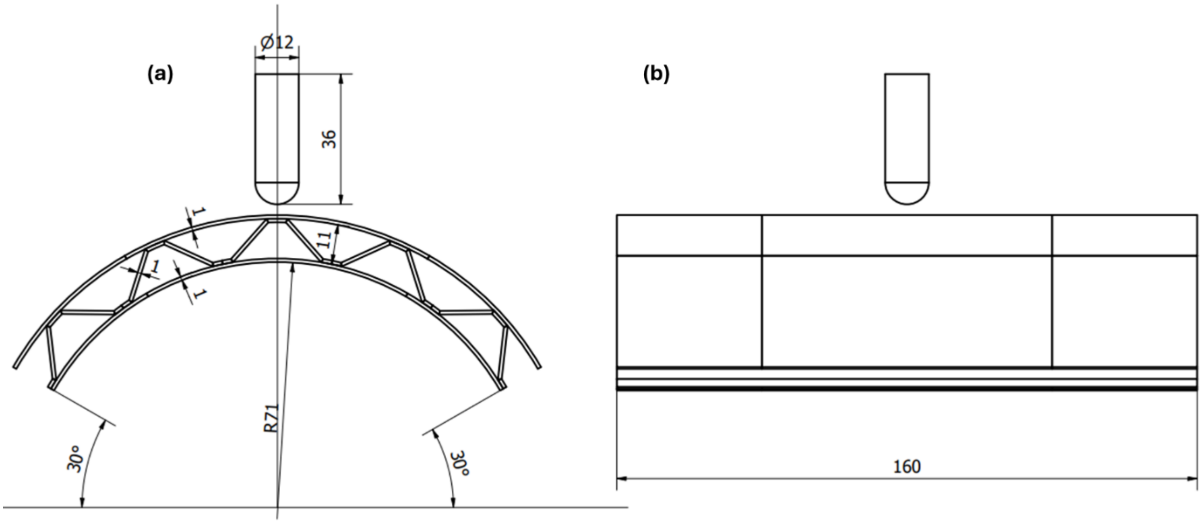

Figure 6. 3D geometry result, (a) Front view, (b) Side view

3. MODEL GENERATION

In the developed program, the model of the curved sandwich structure is created by filling in several parameters in GUI (see **Figure 4**). These parameters refer to **Figure 5a** and **Figure 5c**. **Figure 5a** shows parameter radius (r), angle (θ), angle open (θ_o), and **Figure 5c** describe details about the parameter based on unit cell, the thickness of the top sheet (t_{us}), the thickness of the bot sheet (t_{bs}), the thickness of the core wall (t_{cw}), and the core height (h_c). **Figure 5b** illustrates how the volume of the face sheet is divided into 9 sections. This division aims to assist in the creation of the structured mesh. Additionally, area number 5 represents the impact zone, where the number of elements can be increased for higher resolution in this region.

As previously mentioned, to ensure the dimensions of the generated 3D model are accurate, Autodesk Inventor CAD software is used to verify that the generated dimensions match the parameters listed in **Table 1**. The generated 3D model is shown in **Figure 6**. The total length of the curved sandwich structure dimensions is 160 mm (**Figure 6b**) following 6 mm radius of projectile and 36 mm projectile length (see **Figure 6a**). In **Figure 6a**., it can be observed that the number of unit cells matches the input parameters value of 5. Additionally, the radius of the curved sandwich structure, the thickness of the face sheets, core wall thickness, and core height, are also consistent with input parameters in **Table 1**, which are 71 mm, 1 mm, 1 mm, 1 mm and 11 mm respectively (**Figure 6a** and **Figure 6b**). This indicates that the development program functions correctly and can be reliably used to generate the geometry of the curved corrugate sandwich structure and its projectile component.

4. MESH GENERATION

In this program, two types of elements are used: hexahedral element and tetrahedral element. The hexahedral element are specifically used for the curved sandwich structure and the tetrahedral element used for the impactor. To configure the meshing, the user needs to input the number of mesh divisions along the $\boldsymbol{n_a}, \boldsymbol{n_b}, \boldsymbol{n_c}, \boldsymbol{n_d}, \boldsymbol{n_l}$ for the unit cell of the corrugated sandwich (**Figure 7a** and **Figure 7b**), and n_a, n_b, n_c, n_d, for face sheets (**Figure 7c**). To facilitate a clear representation of the mesh distribution along the variable, the meshing configuration for the impactor is controlled by specifying the minimum and maximum element size, ensuring an appropriate level of refinement in the mesh generation process.

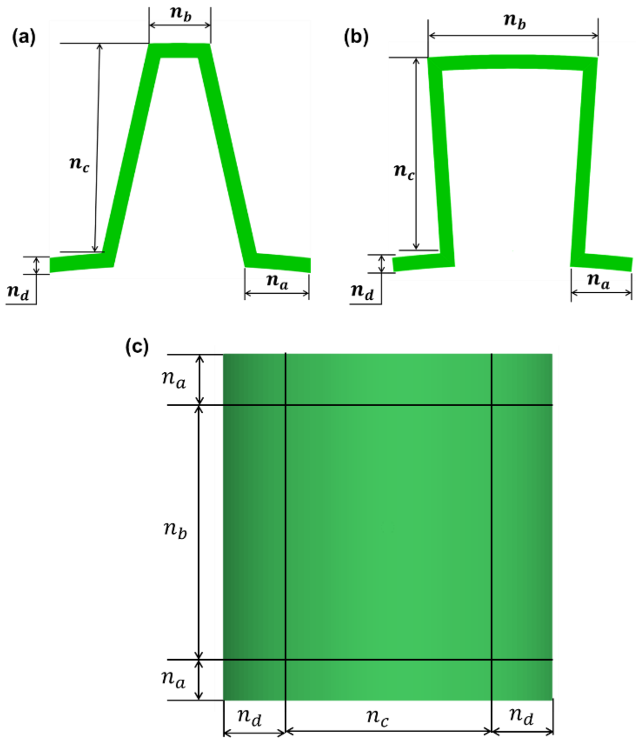

Figure 7. Assembly result of corrugated sandwich structure with trapezoidal core shape and conical impactor; (a) Mesh setting for trapezoidal shape, (b) Mesh setting for rectangular shape, (c) Mesh setting for face sheet.

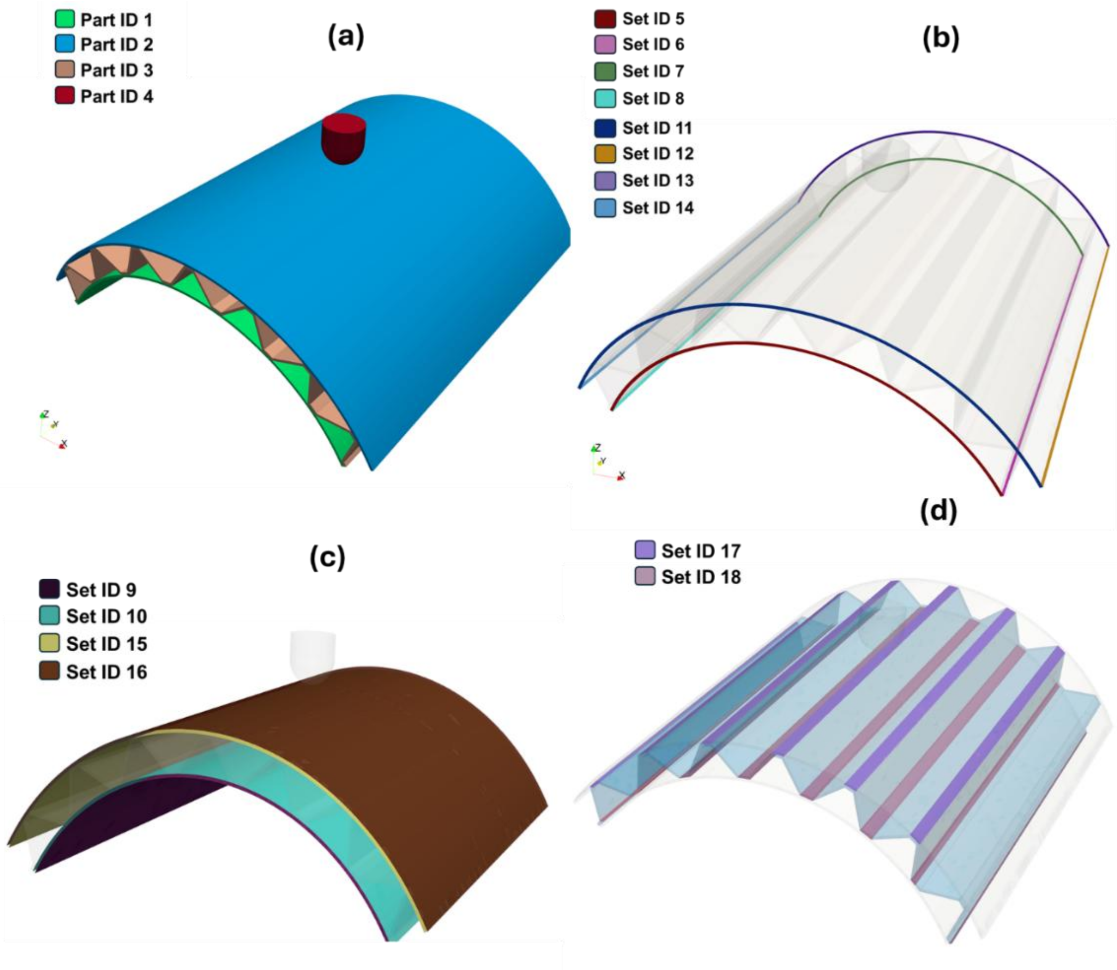

Figure 8. Division of part ID and set ID for organizing the model components; (a) Part ID, (b) Set ID for the side portion of the face sheet, (c) Set ID for the top and bottom portions of the facesheet, (d) Set ID for interaction between the core and facesheet

5. SOLVER INTEGRATION

In general, when using OpenRadioss solver, at least two files are required for the simulation: the starter input (.rad) and engine input (.rad). The starter input file contains information on various parameters, such as material properties, boundary conditions, initial conditions, contact and more. On the other hand, the engine input file contains details related to simulation setting, total simulation time, time step used, and output variables such as damage, displacement, stress, and strain. For more details on the use of the starter and engine input files, please refer to the OpenRadioss reference guide.

In order to facilitate the integration of the finite element model with OpenRadioss solver, it is beneficial to assign set numbers to each part and surface of the models (see **Figure 8**). The set numbers are used to simplify the manage, and definition part, boundary conditions, initial conditions and contact, and it is important to note that the set number assigned is the same across all model crated.

```
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
#-  5. BOUNDARY CONDITIONS BOT SHEET:
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|

/GRNOD/SURF/1
BOT SHEET FRONT          <------   Extract Node from Set ID 5
1

/GRNOD/SURF/2
BOT SHEET RIGHT          <------   Extract Node from Set ID 6
2

/GRNOD/SURF/3
BOT SHEET BACK           <------   Extract Node from Set ID 7
3

/GRNOD/SURF/4
BOT SHEET LEFT           <------   Extract Node from Set ID 8
4

/GRNOD/GRNOD/5
FIX BOT SHEET SET NODE
         1         2         3         4     <------   Group Node from Extract Node

/BCS/1
FIX BOT SHEET
#  Tra rot    skew_ID  grnod_ID
   111 111          0         5    <------   Boundary Conditions
```

Figure 9. Example defining boundary conditions in OpenRadioss solver on the bot sheet

```
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
#-  4. PARTS:
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|

/PART/1
BOT SHEET
         2         1    <------   Element and material properties for part ID 1

/PART/2
TOP SHEET
         2         1    <------   Element and material properties for part ID 2

/PART/3
CORE
         2         1    <------   Element and material properties for part ID 3

/PART/4
PROJECTILE
         1         2    <------   Element and material properties for part ID 4
```

Figure 10 . Example defining Part in OpenRadioss solver

```
#---1----|----2----|----3----|----
#include file_mesh.inc    <----  Import mesh file
#include rigid_body.inc   <----  Import rigid body
#---1----|----2----|----3----|----
```

Figure 11. Import mesh file and rigid body in starter file

There are at least four set numbers which represent Part ID defining each component of the model. Part ID 1, Part ID 2, Part ID 3, and Part ID 4 correspond to the bot sheet, core, top sheet, and projectile, respectively (see **Figure 8a**). Set ID 5 to Set ID 8 define the front, right side, back, and left side of the bot sheet component. Similarly, Set ID 11 to Set ID 14 define the corresponding parts of the top sheet component (see **Figure 8b**). Set ID 9, Set ID 10, set ID 15 and Set ID 16 refer to top and bottom surfaces of the top sheet and bot sheet component, respectively (see Figure 8c). Finally, Set ID 17 and 18 refer to the surfaces interacting between the top sheet and bottom sheet with the core component, respectively (see **Figure 8d**).

To make the use of the Set ID and Part ID easier to understand as explained earlier, a simple example is provided in Figure 9. This figure shows an example of how to define boundary conditions in

OpenRadioss format. The block format used for this purpose can be found in the refrence guide of the OpenRadioss solver. To create the block format, editors Notepad ++ can be used. In the example, the nodes from front, right side, back and left side of the bot sheet component are extracted and the signed as fixed. Similarly, the initial condition for the projectile component is defined in the same manner. To define parts in the OpenRadioss solver (**Figure 10**), it is sufficent to match it with previously assigned Part ID (see **Figure 8a**).

In the integration with OpenRadioss solver, this program separates the mesh file (.inc), starter input (.rad), engine input (.rad) and rigid body (.inc). This separation aims to facilitate the management of input data and simulation process. For the simulation, these separated files must be in the same folder. The mesh file (.inc) contains information such as node, coordinate, element, Part ID and Set ID, which is automatically generated when the model is created. In this development program, we use an in-house code to convert the mesh file (.msh) to mesh file OpenRadioss (.inc) so that it can be read by OpenRadioss solver. The rigid body file (.inc) contains information about the rigid body including the mass of the projectile. To perform the simulation, simply import rigid body file (.inc) and mesh file (.inc) into starter file (.rad) (**Figure 11**).

6. VALIDATION AND TESTING SCENARIOS

To ensure that the computer program functions correctly, validation and scenarios testing are perfromed. Additionally, to simplify the program usage, illustration using diagram flowchart can be seen in **Figure 12**. There are several stages involved, such as creating directory folder, preparing starter and engine input, setting geometry of the curved sanwich structure, setting geometry and mass of the projectile, and mesh setting. Afterward, to assess the mesh and its quality, the program provides a skewness quality check, where the pyVista library (Sullivan & Kaszynski, 2019) used to view the mesh result and check quality (**Figure 13a** and **Figure 13b**). It is important that the skewness quality check is performed only for hexahedral element, meaning that the mesh quality is evaluated solely for the curved sandwich structure, while the quality of element in the projectile is ignored.

The validation between the simulation and experimental result is referred to the previous study, in which a curved sandwich structure is tested with trapezoidal core shape (Yang et al., 2020). The damage result from simulation is shown in **Figure 13c**. The comparison result between the simulation and experiment is shown in **Figure 13d**. It is evident that the result from simulation closely match the experimental impact force pattern. Therefore, based on simulation result it can be concluded that developed program is capable of performing simulation, and the generated model is acceptable.

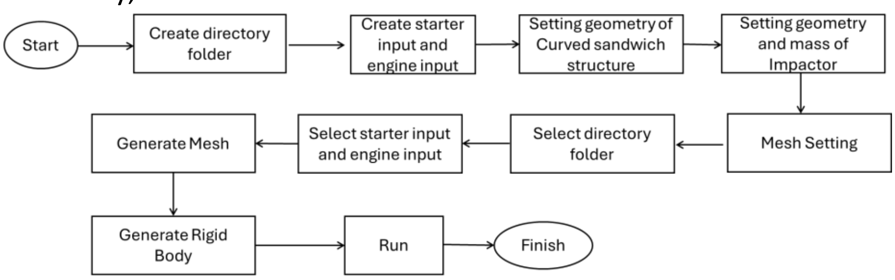

Figure 12. Flow chart usage of program

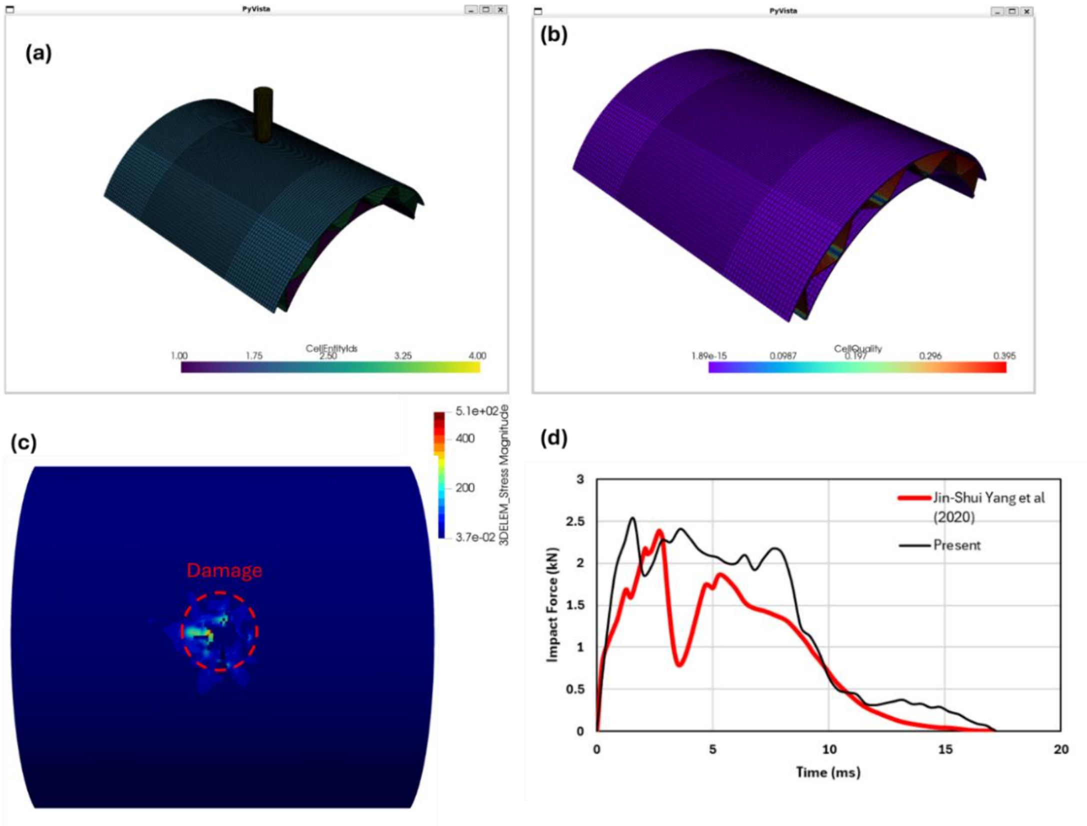

Figure 13. Mesh viewer and validation, (a) Mesh viewer, (b) Mesh quality, (c) Damage simulation, (d) Validation with experiment.

7. RESULT AND DISCUSSION

After conducting the simulation and validation, the next step is to assess how quickly the developed program can generate the finite elemen model. This process aims to evaluate the efficiency in model creation, considering the various design variation that need to be processed. The speed of model creation is compared with conventional method to assess advantages of developed automated program. The model to be tested can be seen in **Table 1**. For the conventional method, software CAD Autodesk Inventor student version is used to create the model, and then export to Gmsh mesh generation for meshing process. The comparation result show that the developed program requires a total time of approximately 73 s, and for conventional method requires a total time of approximately 480 s (see **Table 2**). This indicates that the proposed program can significantly reduce the time requires to create finite elemen model.

Table 2. Comparation consuming time generate FE model.

Method	Value	Unit	Description
This research	± 73	s	Without CAD – Generate Meshing.
Commonly used	± 480	s	Using CAD – Export step file – Generate Meshing.

Next, regarding the advantages and disadvantages compared to the lastest GUI OpenRadioss solver, the comparison result is shown in **Table 3**. From the comparison result, it can be seen that the developed progam offers signifficant advantages especially as a tool for simulating curved sandwich structure with rigid body projectiles. The limitation of the program is also evident, with the main limitation being that it can only be used for simulating curved sandwich structure.

Additionally, it is currently only compatible with linux OS.

Next, regarding mesh quality, as previously explained, we use skewness calculation to assess the quality of the meshing result. The result of this mesh quality can be seen in **Figure 14e**, where it is shown that the mesh quality is very good. Additionally, simulation were performed with variations in the core structure shapes, including rectangular and trapezoidal (**Figure 14c** and **Figure 14 d**) and simulations with projectile shape variations, including conical, spherical, and flat (**Figure 14a, Figure 14b** and **Figure 14d**). It can be observed that the damage was succsessfully simulated, indicating that the simulation ran smoothly without error and bug.

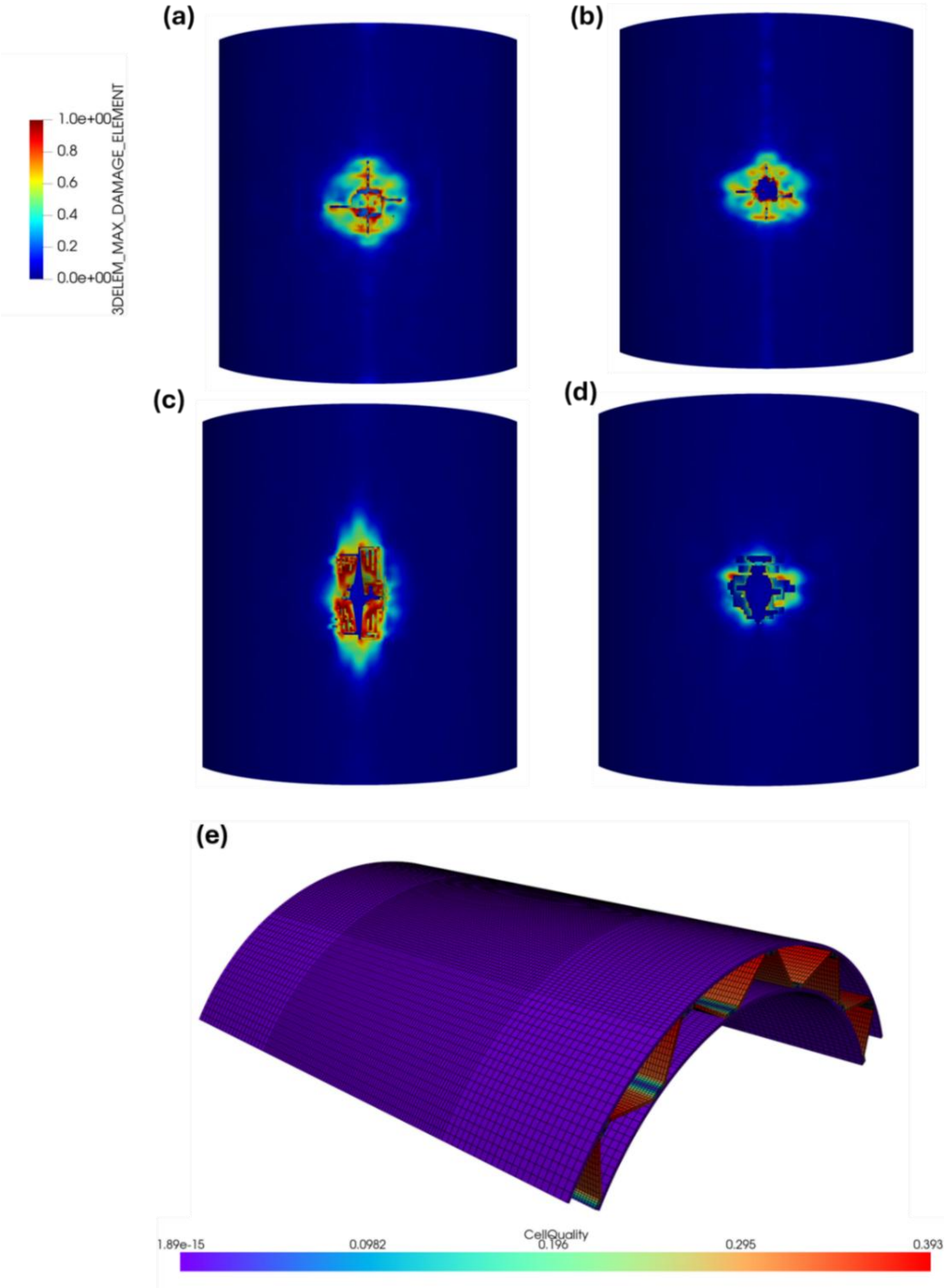

Figure 14.Mesh quality and damage simulation with different projectile and core shape, (a) Flat projectile, (b) Conical, (c) Rectangular, (d) Trapezoidal with Hemispherical impactor, (e) Mesh quality.

Table 3. Comparation with other GUI

GUI	Advantages	Disadvantages
Proposed GUI	• No CAD required to create geometry, and FE model generation is very fast. • Mesh viewer is available via pyVista. • Quality mesh checking is available via pyVista. • Integrated with mesh generation via Gmsh.	• Limited use for projectile impact on curved sandwich structures with a rigid body projectile. • Limited to OpenRadioss solver. • Limited to hexahedral and tetrahedral elements. • Currently, it can only be run on Linux OS.
Latest GUI OpenRadioss	• More general use. • Compatible with another solver such as LS-Dyna and Abaqus. • Compatible with Windows or Linux.	• Requires CAD software for creating geometry, followed by conventional meshing. • Mesh viewer is not available • Requires external software such as ParaView to view mesh results. • Quality mesh checking is not available. • Not integrated with mesh generation • External mesh generation is needed such as Gmsh.

8. CONCLUSION

In this study, a self-developed parametric modeling tool was introduced to automate the creation of CAD and FEM models for curved corrugated sandwich structures. The tool has been proven to be effective in simplifying the modeling process, where users no longer need to model the geometry with CAD, but simply input the required parameters. The program takes approximately ±73 s to generate the finite element model, while the conventional method takes about ±480 s. This indicates that the developed program significantly reduces modeling time.

Additionally, the developed program includes a mesh viewer, mesh quality check, and successful integration with the OpenRadioss solver. This integration streamlines the simulation process and enhances the overall efficiency of model generation and analysis. The current version of the tool is limited to core shapes (trapezoidal and rectangular) and projectile shapes (flat, spherical, and conical). The tool is primarily designed for curved corrugated sandwich structures under projectile impact only. Future improvements could include automatic FE model generation and optimization, such as using the Taguchi method.

REFERENCES

Bozkurt, I. (2024). Effect of geometric configurations and curvature angle of corrugated sandwich structures on impact behavior. Polymer Composites. https://doi.org/10.1002/pc.29064

Geuzaine, C., & Remacle, J.-F. (2009). Gmsh: A 3-D finite element mesh generator with built-in pre-and post-processing facilities Gmsh: a three-dimensional finite element mesh generator with built-in pre-and post-processing facilities. International Journal for Numerical Methods in Engineering, 79(11). https://doi.org/10.1002/nme.2579ï

Hashemi, M., Hatami, O., & Tajbakhsh, M. R. (2024). Investigation of the performance of the structure and energy absorption in sandwich panels of PLA/TPU manufactured by the FFF technique. Proceedings of the Institution of Mechanical Engineers, Part L: Journal of Materials: Design and Applications, 238(3), 416–429. https://doi.org/10.1177/14644207231188827

Peterson, E., & Kaur, B. (2024). Printing Compound-Curved Sandwich Structures with Robotic Multi-Bias Additive Manufacturing. In Computational Design and Robotic Fabrication: Vol. Part F2072 (pp. 526–536). Springer. https://doi.org/10.1007/978-981-99-8405-3_44

Sullivan, C., & Kaszynski, A. (2019). PyVista: 3D plotting and mesh analysis through a streamlined interface for the Visualization Toolkit (VTK). Journal of Open Source Software, 4(37), 1450. https://doi.org/10.21105/joss.01450

Sun, G., Huo, X., Wang, H., Hazell, P. J., & Li, Q. (2021). On the structural parameters of honeycomb-core sandwich panels against low-velocity impact. Composites Part B: Engineering, 216. https://doi.org/10.1016/j.compositesb.2021.108881

Tarlochan, F. (2021). Sandwich Structures for Energy Absorption Applications: A Review. Materials, 14(16). https://doi.org/10.3390/ma14164731

Um, H. J., Rho, H. J., Jeon, N. H., Shin, J. H., & Kim, H. S. (2025). Mechanical performance of novel curved sandwich structures featuring 3D printed continuous carbon fiber/polyamide 6 composite corrugated core with rail interlocking. Composites Part B: Engineering, 295. https://doi.org/10.1016/j.compositesb.2025.112222

Wang, G., Li, Z., Lu, Z., & Liu, J. (2024). Local impact resistance of curved sandwich panel with Miura-ori foldcore. Engineering Structures, 300. https://doi.org/10.1016/j.engstruct.2023.117232

Yang, J. S., Zhang, W. M., Yang, F., Chen, S. Y., Schmidt, R., Schröder, K. U., Ma, L., & Wu, L. Z. (2020). Low velocity impact behavior of carbon fibre composite curved corrugated sandwich shells. Composite Structures, 238. https://doi.org/10.1016/j.compstruct.2020.112027

Zhou, L., Miller, J., Vezza, J., Mayster, M., Raffay, M., Justice, Q., Al Tamimi, Z., Hansotte, G., Sunkara, L. D., & Bernat, J. (2024). Additive Manufacturing: A Comprehensive Review. In Sensors (Vol. 24, Issue 9). Multidisciplinary Digital Publishing Institute (MDPI). https://doi.org/10.3390/s24092668