



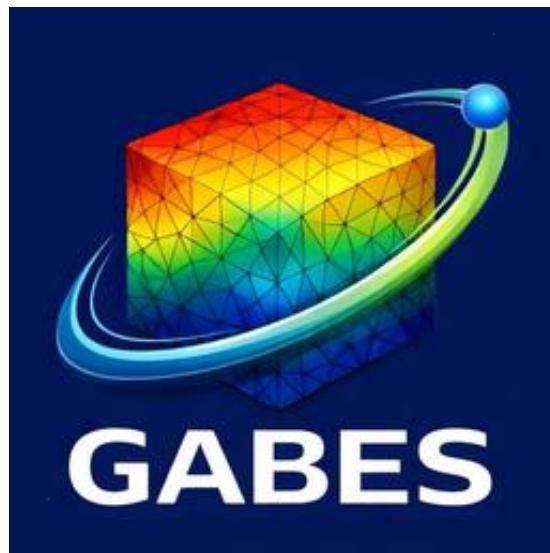
المدرسة الوطنية المتعددة التقنيات
Ecole Nationale Polytechnique



GABES

Graphical & Accelerated Boundary Element Solver

User's Guide



SI HADJ MOHAND Rahim
BELKACEMI Yacine
RECHAK Said

NATIONAL POLYTECHNIC SCHOOL | ALGIERS, ALGERIA | JANUARY 2026

1 INTRODUCTION

GABES (Graphical & Accelerated Boundary Element Solver) is an open-source application based on the Boundary Element Method (BEM), accelerated using an \mathcal{H} -Matrix library, for solving three-dimensional linear elastostatic problems in isotropic, homogeneous media. The application runs exclusively on Windows and provides a user-friendly graphical interface with an interactive 3D environment. Users can import meshed 3D models, define numerical and physical parameters, apply boundary conditions interactively, and launch simulations. GABES also includes post-processing tools for graphical visualization of results and the generation of tabulated reports.

1.1 Download and installation

GABES is available as an open source package in the GitHub platform, with the repository accessible via the link <https://github.com/rahimsihadjmh/GABES.git>

2 PRESENTATION OF THE APPLICATION

2.1 Graphical parts

2.1.1 The Mainframe Window

The **Mainframe** window serves as the container for all graphical components of the application, as illustrated in Figure 01. It includes:

- **Client Area:** the primary workspace for 3D visualization and interaction;
- **Ribbon:** the main toolbar for commands and tools;
- **View Toolbar:** a toolbar for commands and tools to manipulate the 3D model;
- **Model Tree:** a hierarchical display of the various objects used during the modeling phase;
- **Information Panel:** displays context-specific data and parameters;
- **Status Bar:** shows the coordinates and the indices of the selected elements and node.

Each of these components will be described in more detail in the following sections.

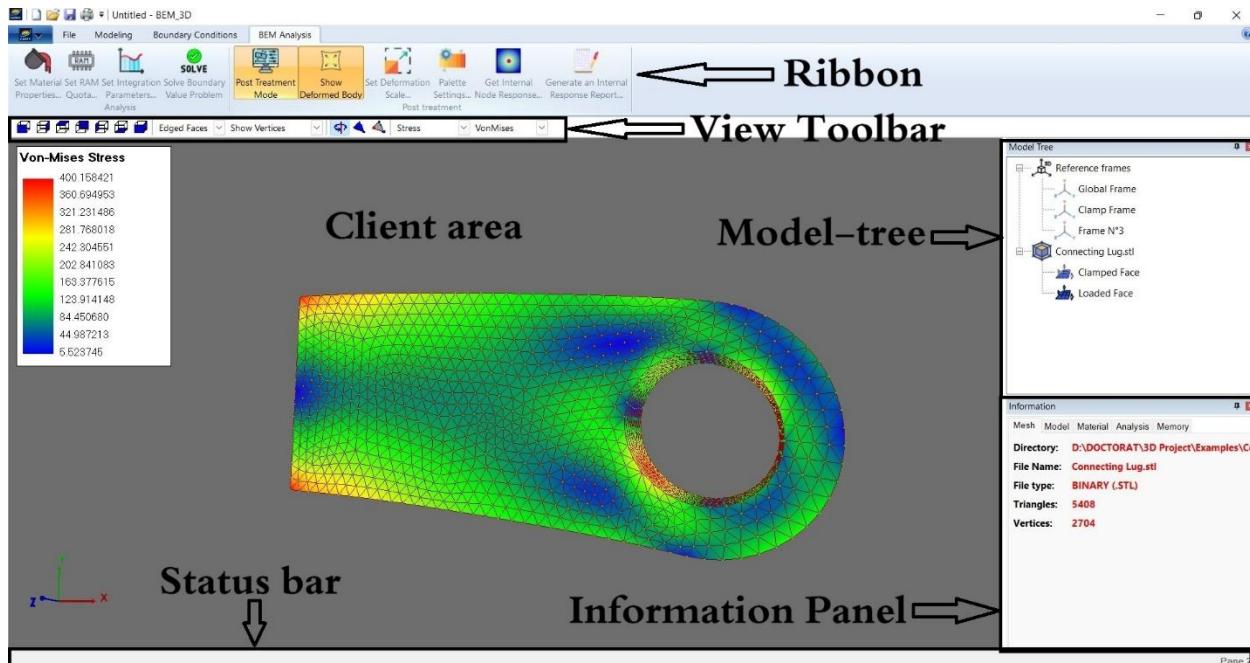


Figure 1: Mainframe Window

2.1.2 The Client Area

The **Client Area** is the primary space for visualizing the 3D model in both the modelling and post-processing phases. It permits rotation of the 3D model and contains a small representation of the three coordinate axes (Oxyz), known as the **Gizmo**.

In the **modelling phase**, the Client Area allows interactive selection of triangular mesh elements, vertices, and DOF nodes, which can be useful for various purposes. It also displays several objects (see Figure 2):

- The imported model mesh
- The created coordinate frames
- The created surface subsets
- The applied boundary conditions (simple/double fixations, displacement vectors, and traction vectors)

In the **post-processing phase** (see Figure 3), it displays:

- Various color-mapped field variables (displacements, stresses, and strains)
- A color palette with the corresponding values
- The deformed and undeformed configuration of the body

The model can be visualized in different output modes, including **solid surface**, **edged surface**, and **wireframe**. The visibility of geometrical mesh vertices and DOF nodes can be toggled on or off. All these view options can be controlled and configured using the **View Toolbar**.

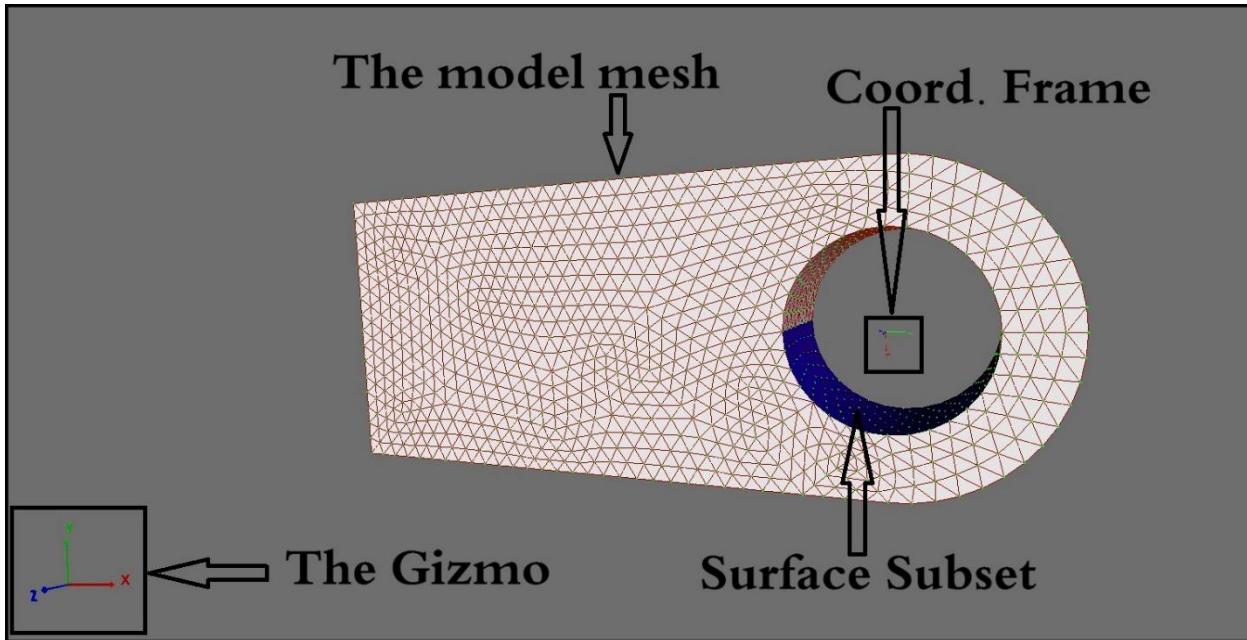


Figure 2: Client Area modeling phase

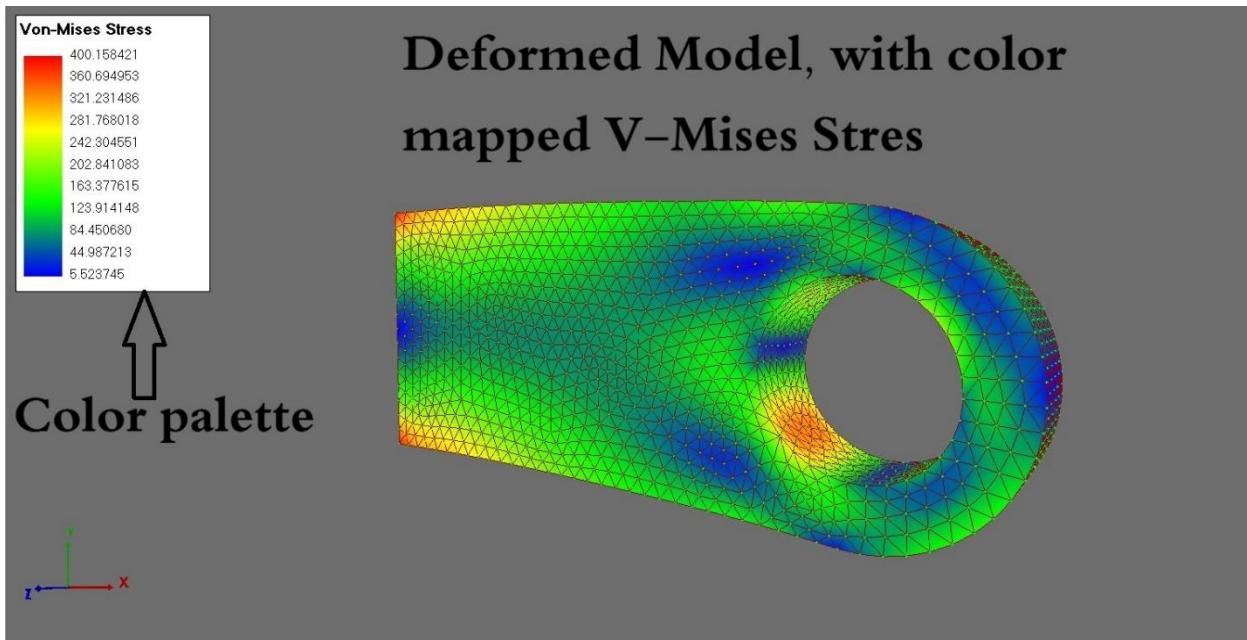


Figure 3: Client Area post-processing phase

2.1.3 The Ribbon

Modern GUI applications feature a **Ribbon**, which replaces the traditional menu and toolbar system. Commands are provided through buttons with larger icons, organized into panels across multiple separate tabs, making them more flexible and easily accessible.

The majority of **GABES**'s commands are embedded in the **Ribbon**, which is organized into four (04) tabs:

2.1.3.1 File tab

Subdivided into two (02) panels:

- **Geometry panel:** Contains two (02) commands for loading and deleting the model mesh
- **GABES File Management panel:** Contains four (04) commands for creating, opening and saving GABES files.

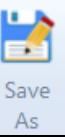


Figure 4: File tab

The **File tab** commands are summarized in Table 1

Table 1: Command buttons of the File tab

Panel	Button	Action
Geometry	Load Mesh	Opens a standard Windows Open File dialog to navigate to the location of the mesh file, which can be in one of two formats: .stl or .obj . The 3D surface mesh is then loaded into GABES
	Delete the Model	Deletes the current model and cleans up all generated data
GABES File Management	New	Creates a new GABES file and may prompt the user to save the current model if modifications have occurred
	Open	Opens an existing GABES file, which may contain all modeling information, including the mesh, reference frames, surface subsets, boundary conditions, and field variable results.

	 Save	Saves the current model to an existing GABES file, or prompts the user to choose a save location if the file has not yet been created
	 Save As	Saves the current model to a new disk location by prompting the user to choose the destination directory.

2.1.3.2 Modeling tab

The tab is subdivided into two (02) panels:

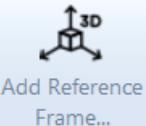
- **Frames panel:** Contains a command for creating new reference frames, which are very helpful during surface subset creation and post-processing report generation.
- **Surface Subsets panel:** Contains three (03) different commands for generating surface subsets, which are mandatory for boundary condition assignment.



Figure 5: Modeling tab

The **Modeling tab** commands are summarized in Table 2

Table 2: Command buttons of the Modeling tab

Panel	Button	Action
Frames		Opens a modeless dialog box that allows the graphical, interactive creation of a new frame by defining the location of the origin point and the orientation of the axes.
Surface Subsets		Creates a new surface subset from the currently selected elements. This button becomes active only when a valid selection exists N.B. Elements can be selected interactively in the 3D model using the mouse cursor, with the CTRL key enabling multiple selections.
		This command opens a standard Windows Open File dialog to navigate to the location of a .obj file describing the subset geometry. This method can be used with meshes created in Autodesk 3ds Max , where individual surfaces can be

		<p>selected and detached using the 3ds Max Detach command. These surfaces can then be exported as Wavefront .obj files, which GABES can read and convert into surface subsets.</p> <p>N.B. This method is widely adopted in the BESLE [Ref] software.</p>
		<p>This command implements the third method in GABES for creating surface subsets. It uses previously defined coordinate frames to map model coordinates into the selected frame within one of three coordinate systems: Cartesian, cylindrical, or spherical. The user can then specify coordinate ranges either by entering intervals manually or by using the provided slider bars, with selections updating graphically in real time. This method is particularly efficient for well-structured geometries with planar, cylindrical, or spherical sections, which is typical for most mechanical parts.</p>

2.1.3.3 Boundary Conditions tab

The tab is subdivided into two (02) panels:

- **Displacement BCs:** Contains commands for assigning various Dirichlet-type boundary conditions, including total encaster (clamping), partial encaster (zero displacement in selected directions), vectorial displacement, and normal displacement. This panel also provides check boxes to toggle the visibility of the created fixations and displacement vectors.
- **Traction BCs:** Contains commands for assigning various Neumann-type boundary conditions, including vectorial traction, normal traction, and pressure. It also provides the **Free Surface** command to release a surface from the previously assigned boundary conditions and restore it to a traction-free state. A checkbox is also available to toggle the visibility of the created traction vectors.

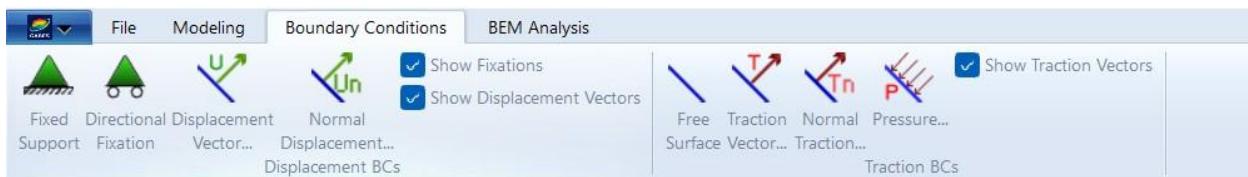
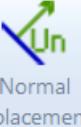
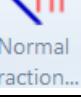
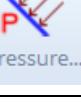


Figure 6: Boundary Conditions tab

These boundary condition assignment tools are only active when a **Surface Subset** is selected in the **Model Tree**. The commands on the **Boundary Conditions tab** are summarized in Table 3.

Table 3: Command buttons of the Boundary Conditions tab

Panel	Button	Action
Displacement BCs	 Fixed Support	Assigns a fully fixed boundary condition (zero displacement in all three directions) to the selected surface subset.
	 Directional Fixation	Assigns zero displacement in the prescribed directions. The user is prompted with a dialog box to select the directions where zero displacement is applied.
	 Displacement Vector...	Assigns prescribed displacements in selected directions. The user is prompted with a dialog box to specify the directions and the corresponding displacement values.
	 Normal Displacement...	Assigns a displacement vector normal to the elements with a prescribed magnitude. The user is prompted with a dialog box to specify the displacement magnitude.
	<input checked="" type="checkbox"/> Show Fixations	Toggles the visibility of the created fixations on and off.
	<input checked="" type="checkbox"/> Show Displacement Vectors	Toggles the visibility of the created displacement vectors on and off.
Traction BCs	 Free Surface	Removes the previously assigned boundary conditions from the selected surface, restoring it to a traction-free state.
	 Traction Vector...	Assigns prescribed tractions in specified directions. A dialog box prompts the user to select the directions and enter the corresponding traction values.
	 Normal Traction...	Assigns a traction vector normal to the elements with a specified magnitude. A dialog box prompts the user to enter the traction magnitude.
	 Pressure...	This command is similar to the Normal Traction command but acts in the opposite normal direction. Positive pressure is applied inward.
	<input checked="" type="checkbox"/> Show Traction Vectors	Toggles the visibility of the created traction vectors on and off.

2.1.3.4 BEM Analysis tab

The tab is divided into two panels:

- **Analysis Panel:** Contains commands for defining material properties, setting numerical integration parameters, managing memory, and launching simulations.
- **Post-Processing Panel:** Contains commands and tools for various post-processing tasks, including enabling or disabling post-processing view mode, showing or hiding the deformed configuration of the body, setting the displacement magnification scale, choosing palette options, and obtaining internal domain responses either for individual points or as tabulated reports for multiple points.

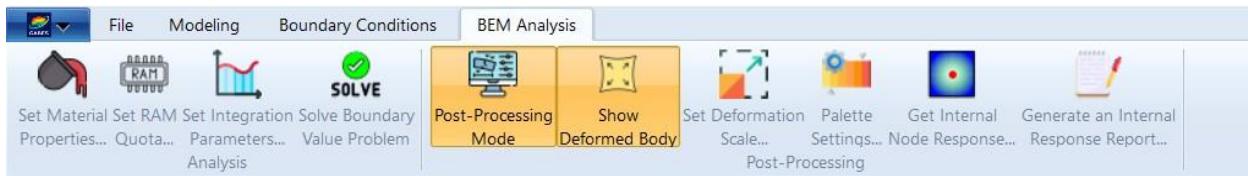


Figure 7: BEM Analysis tab

The commands on the **BEM Analysis tab** are summarized in Table 4

Table 4: Command buttons of the BEM Analysis tab

Panel	Button	Action
Analysis	Set Material Properties...	Prompts the user with a dialog box to specify the material parameters. For linear elasticity, these include the Young's modulus E , the shear modulus μ , and the Poisson's ratio ν . By default, the parameters are set to the values for structural steel ($E = 200$ GPa, $\nu = 0.3$).
	Set RAM Quota...	Prompts the user with a dialog box to specify the reserved RAM memory quotas for the BEM system matrices. If a matrix exceeds its allocated RAM quota, the remaining data is stored on the hard drive using the Out-Of-Core (OOC) storage method.
	Set Integration Parameters...	This command opens a dialog box for configuring various integration parameters, including whether to use the Rigid Body method, selecting the cubature rule, or enabling an adaptive integration criterion.
	SOLVE Solve Boundary Value Problem	This command is the most important operation in the application. Once the model is fully prepared and all parameters are correctly defined, the user clicks this button to solve the problem. Before execution, a dialog box prompts the user to select

		the \mathcal{H} -matrix error tolerance and the solver to use (H-LU, BiCGStab, or GMRES).
Post-Processing	 Post-Processing Mode	This button enables the Post-Processing View mode
	 Show Deformed Body	This button becomes active when the Post-Processing view mode is selected. After the problem is solved, this command allows the deformed body to be displayed by applying the computed displacement field to the mesh vertices using a uniform magnification scale.
	 Set Deformation Scale...	This button opens a dialog box for setting the deformation magnification scale. It becomes active only in Post-Processing view mode and after selecting the previous Show Deformed Body button
	 Palette Settings...	Opens a dialog box for configuring the number formatting displayed in the Color Palette. Tick values can be shown in either floating-point or scientific (exponential) notation, with a user-defined number of decimal digits.
	 Get Internal Node Response...	Opens a dialog box for computing specified field variables at an internal domain point or on a boundary point. The user enters the spatial coordinates of the point and selects the variables to be evaluated (displacements, stresses, or strains). The results are displayed directly in the dialog box.
	 Generate an Internal Response Report...	This command serves the same purpose as the previous one. However, instead of specifying a single point, the user is prompted to select a point cloud (multiple points), which can be generated automatically or imported from an input coordinate file. After computation, the results are exported to a tabulated report file in CSV format, readable by spreadsheet programs such as Microsoft Excel.

2.1.4 Dockable panels

Dockable panels are user interface components that can be attached to the edges of the main application window or displayed as independent floating windows, allowing users to customize the workspace layout according to their preferences and workflow. GABES has two distinct dockable panels, the Model-Tree panel and the Information panel. (See Figure 8).

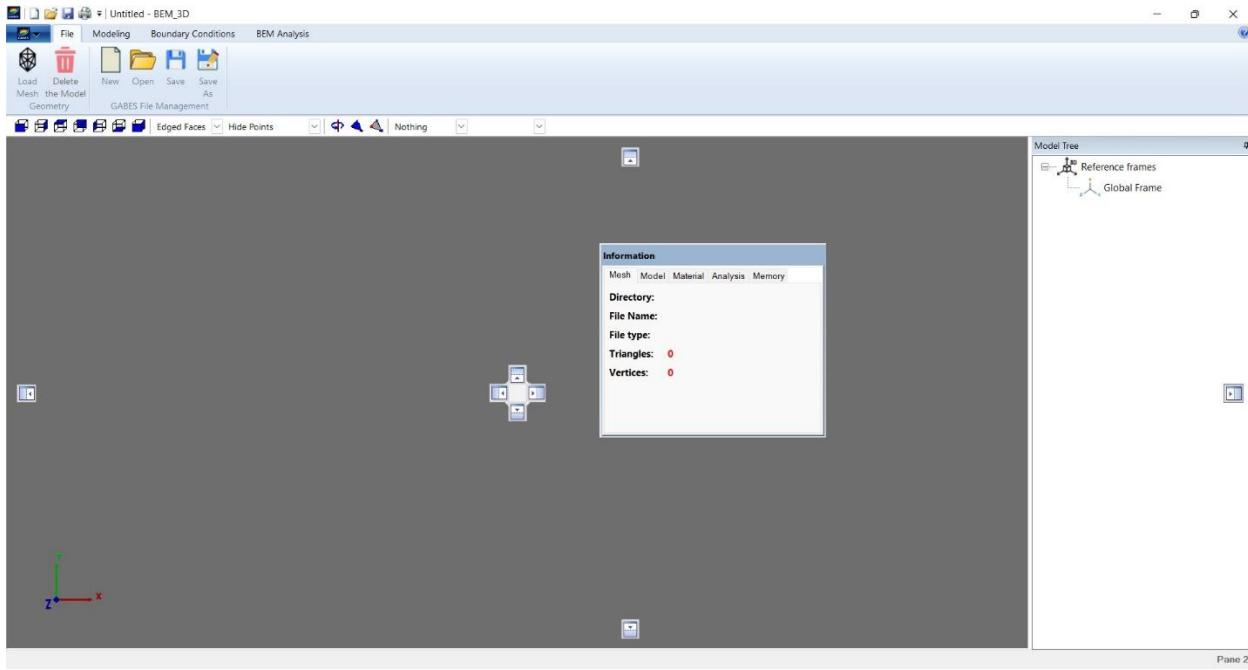


Figure 8: Dockable panels

2.1.4.1 The Model-Tree panel

The **Model Tree** is a floating and dockable panel that can be attached to any edge of the main frame window (see Figure 8). However, since the model hierarchy typically grows in the vertical direction, it is more practical to dock this panel on the **left or right side** of the main frame, where greater vertical space is available and navigation becomes more convenient.

The Model Tree panel displays the objects created during the modeling phase, including reference coordinate frames, the imported mesh, and the associated surface subsets. The panel is organized as a hierarchical tree structure composed of two (02) root nodes:

- **Reference Frames root:** contains all user-defined reference coordinate frames as well as the default global frame, which is permanently associated with the model's center of gravity and aligned with the center of the Client Area.
- **Mesh root:** becomes available once a mesh is loaded and adopts the name of the mesh file. All generated surface subsets are organized as child nodes of this root.

The Model Tree facilitates the modeling workflow by enabling easy selection and modification of reference frames. It also allows the selection of surface subsets, which activates the boundary-condition tools and simplifies the assignment and modification of boundary conditions.

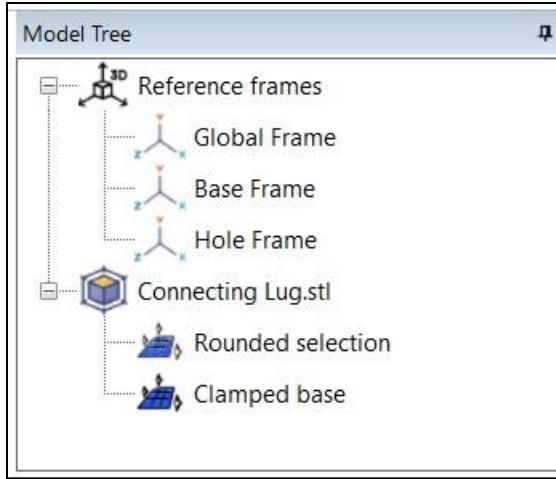


Figure 9: The Model-Tree panel

The objects displayed on the **Model-Tree panel** are summarized in Table 5

Table 5: The Model-Tree panel objects

Object	Description
Reference frames	Root node containing all reference coordinate frames defined in the model.
Global Frame	The global reference coordinate frame automatically created by the application and permanently associated with the model's center of gravity.
Hole Frame	A reference coordinate frame created by the user for custom positioning, orientation, or boundary-condition definition.
Connecting Lug.stl	Root node representing the imported mesh; its name is extracted from the mesh file title.
Clamped base	A surface subset created by the user from the loaded mesh, used for selection and boundary-condition assignment.

2.1.4.2 The Information panel

The **Information panel** is a floating and dockable panel that can be attached to any edge of the main frame window (see Figure 8). It is particularly convenient to dock it below the Model Tree, as this preserves more space for the main view area.

This panel shows essential geometrical and numerical information, along with memory storage requirements. This helps the user understand the size of the model and the system matrices, making it easier to adjust simulation settings efficiently. (See Figure 10)

GABES uses \mathcal{H} -Matrix techniques to speed up the solution of the linear system. However, the boundary condition handling requires the matrices to be fully assembled first. Once the system is ready, it is converted into an \mathcal{H} -Matrix format to take advantage of fast solvers. This approach keeps the system assembly simple while still benefiting from efficient computation during the solution phase.

Because the full matrices are used during assembly, the memory requirements can be high. To manage this, GABES uses an **Out-of-Core** storage strategy, storing part of the data on disk to avoid overloading the RAM. The **Information Panel** shows memory estimates to help the user decide how much RAM to allocate to each matrix. The RAM allocation is controlled via the “**Set RAM Quota...**” button under the **BEM Analysis** tab of the Ribbon. This method allows simulations of relatively large meshes while balancing memory use and performance.

Information				
Mesh	Model	Material	Analysis	Memory
Total Requirement:	13.93 Gb			
Model Requirement:	2.85 Gb			
[Q] Requirement:	6.41 Gb			
[R]/[A] Requirement:	2.33 Gb			
Fixed RAM Quota :	28.00 Gb			
[Q] Storage RAM/OOC:	6.41 Gb / 0 byte			
[R] Storage RAM/OOC:	2.33 Gb / 0 byte			
[A] Storage RAM/OOC:	2.33 Gb / 0 byte			

Figure 10: Infomration panel

The various information is displayed in five (05) separated tabs:

2.1.4.2.1 Geometry tab

Displays mesh related information, like the location directory of the mesh file, the file title, the file type, the number of triangles and the number of vertices. (See Figure 11).

Mesh	Model	Material	Analysis	Memory
Directory:	D:\DOCTORAT\3D Project\Examples\Connecting Lug			
File Name:	Connecting Lug.stl			
File type:	BINARY (.STL)			
Triangles:	5408			
Vertices:	2704			

Figure 11: Information panel - Mesh tab

The information displayed under the **Mesh tab** is summarized in Table 6

Table 6: The information displayed under the Mesh tab

Information field	Description
Directory	The current hard disk directory where the mesh file is located.
File name	The name of the mesh file.
File type	The mesh file format, which can be either STL (binary or ASCII) or OBJ (ASCII).
Triangles	The total number of triangles in the mesh.
Vertices	The total number of vertices in the mesh.

2.1.4.2.2 Model tab

Displays model related information, like the discretization method, the number of DOF vertices which may differ from the number of the geometrical mesh vertices, the number of continuous and discontinuous elements and the resulting number of the algebraic DOFs which is three (03) times the number of DOF vertices due to the three (03) spatial directions. (See Figure 12)



Figure 12: Information panel - Model tab

The information displayed under the **Model tab** is summarized in Table 7

Table 7: The information displayed under the Model tab

Information field	Description
Disc. Type	The discretization method used, which can be Continuous, Discontinuous, or Disc. Sharp Edges. This corresponds to purely continuous elements, purely discontinuous elements, or automatically placed discontinuous elements at sharp edges and corners.
DOF Vertices	The number of degree-of-freedom (DOF) nodes, which may differ from the number of mesh vertices when discontinuous elements are present.
Cont. Elements	The total number of continuous elements in the model.
Disc. Elements	The total number of discontinuous elements in the model.
Algebraic DOFs	The resulting number of algebraic degrees of freedom, calculated as the number of DOF vertices multiplied by the three spatial dimensions. This value is important because it determines the overall size of the numerical system, including both matrices and vectors.

2.1.4.2.3 Material tab

The panel displays three material parameters: Young's modulus, shear modulus, and Poisson's ratio. As shown in Figure 13, Young's and shear modulus are shown as unitless values, because GABES is a unitless application (similar to ABAQUS and BESLE). The actual units of traction and stress depend on the user's choice of length units when creating the mesh. For example, if the user interprets the geometry in meters, traction is expressed in Pascals (Pa); if in millimeters, traction is expressed in Megapascals (MPa).



Figure 13: Information panel - Material tab