AFP Engine

GUIDE DOCUMENT
Ver. 1.23-a

Aug 2023

Confidential

About

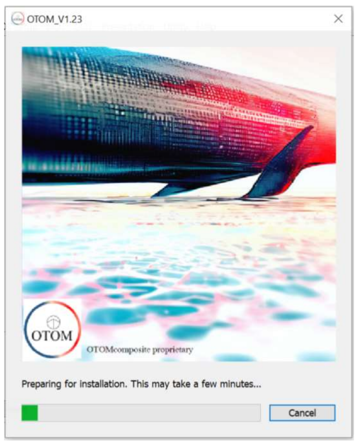
- **What is OTOM?** OTOM AFP engine software is a dedicated simulation software from the OTOM composite for Automated Fiber Placement technology.
- **Why was OTOM developed?** There was a lack of simulation tools for AFP technology when the developments started in 2016. OTOM software was initially developed through the EU ambliFibre project as the adaptive model-based control for laser-assisted fiber-reinforced tape winding of composite pipes and vessels. It was the first software design, implementation, and validation within the project ambliFibre (H2020-EU). OTOM filled this gap by providing software and services for AFP manufacturers and researchers across the world.
- **How OTOM Software works?** OTOM works based on the numerical techniques FEM, and FDM which were tailored for AFP technology. It has built-in boundary conditions and characteristics of the CF/GF composites, especially for the AFP process. OTOM was developed with an efficient algorithm that makes it possible to run it on every laptop or PC.
- **What benefits OTOM offers?** OTOM users can improve their knowledge and process efficiency by obtaining temperatures, Crystallinity, or Energy distribution for the process of manufacturing composites. Users can get the software or request Engineering services from the software.

**OTOM software suits first became available to the public in the Summer of 2023*

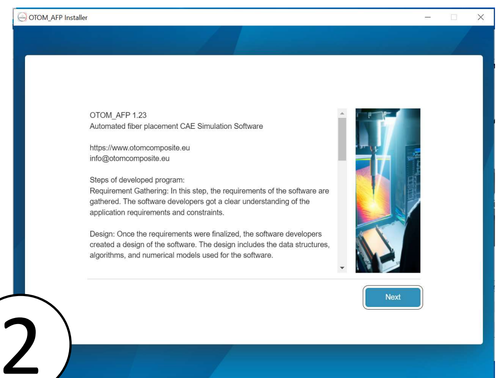
Installation

The installation of OTOM software is rather straightforward. You must click on Next and select the source directory for the installing software and the installation will be finished, You might need an internet connection to download the necessary libraries.

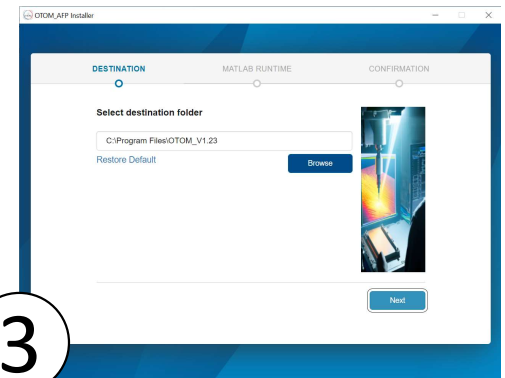
1



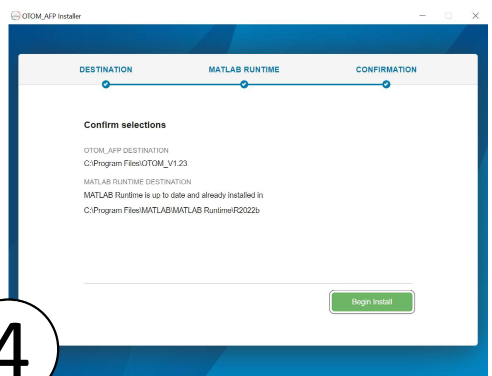
2



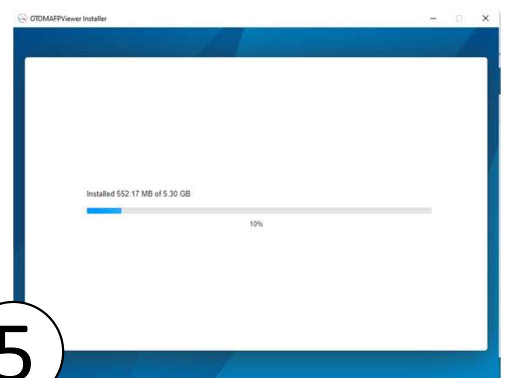
3



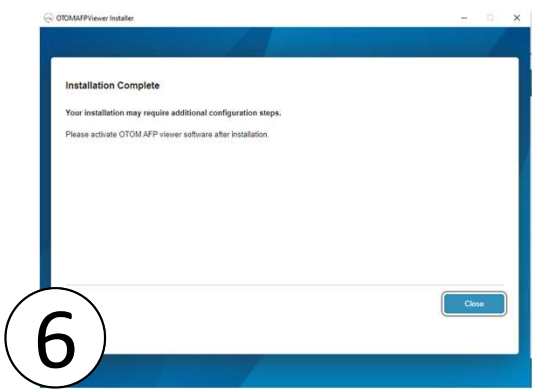
4



5



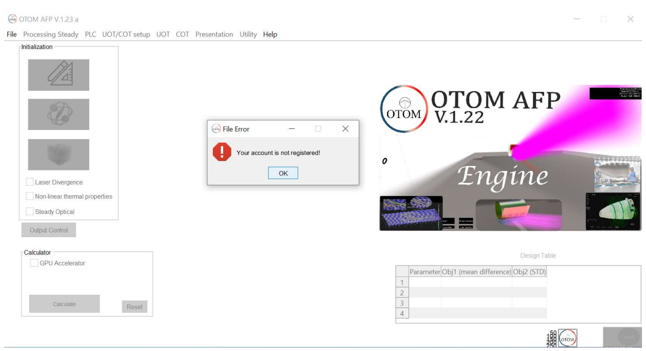
6



Account and licensing

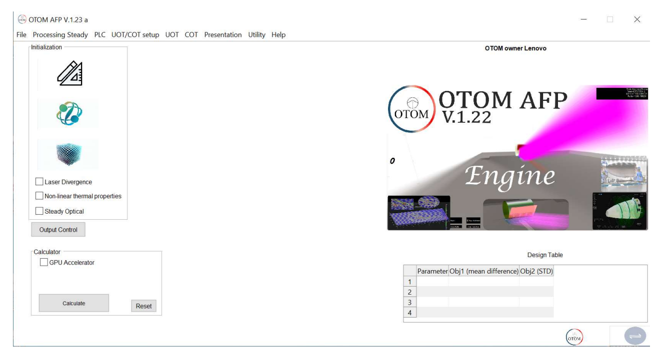
After installation of OTOM AFP software, the main window is locked. Your device should be registered in OTOM composite server to unlock the Software functions.

- To unlock the program click *File> Account.* Your device tries to connect with the OTOM license server and validate your machine. You should write the account name of your company.
- In order to register your machine: *file> Share.* Your MAC address info will be sent to the OTOM server via your Email Microsoft Outlook application. You can check your Email to see the shared info. You can also send them manually to OTOMcomposite by Email.



>>

After License verification



Definitions of objects

- **Tooling (liner)**: this part is a geometry that exists from the beginning of the AFP process, the tapes are placed on them. Every AFP process consists of several tooling, referred to as tooling1, tooling2, tooling3…. The lower number is always at the bottom side in the UOT model.
- **Tape**: This part is a geometry that will be made during the process. It has geometrical evolution. The elements and nodes of placed tapes are called "plied tape" in AFP viewer. Like tooling, the numbering is done as Tape1, Tape2, Tape3 …. Where the lower number comes at the lower position in the UOT model.
- **Roller**: The roller consolidates hot tape on the substrate. This part has contact with the incoming tape and presses the incoming tape on the tooling in the consolidation phase.
- **Substrate**: This part refers to the upper surface of the last layer of the tooling. It could be plied tape or upper tooling in the UOT model.
- **Laser**: Laser is defined as a 2D cross-section area in a 3D space where the surface meshes. This part is the source of the energy which the rays are emitted from.
- **Nip-point:** This is the geometrical location on the tape and the substrate where two parts are connected. The intersection is the line where the consolidation starts
- **Tape-Rack (Roller \ blade):** This part is optional in front of the roller only in UOT model.

Confidential

Definitions of process inputs/outputs

Inputs:

- **Geometrical parameters:** these include all sizes and positions necessary to build up process models such as thickness, relative positions of objects, etc.
- **Process parameters:** these include all necessary properties.
  (thermal\optical\mechanical characteristics, conditions, …) to build up process models such as conductivity, refractive index, laser power, etc.
- **Computational parameters:** All discretization values to be employed in the model performance such as the number of laser rays, thermal nodes, elements, etc.

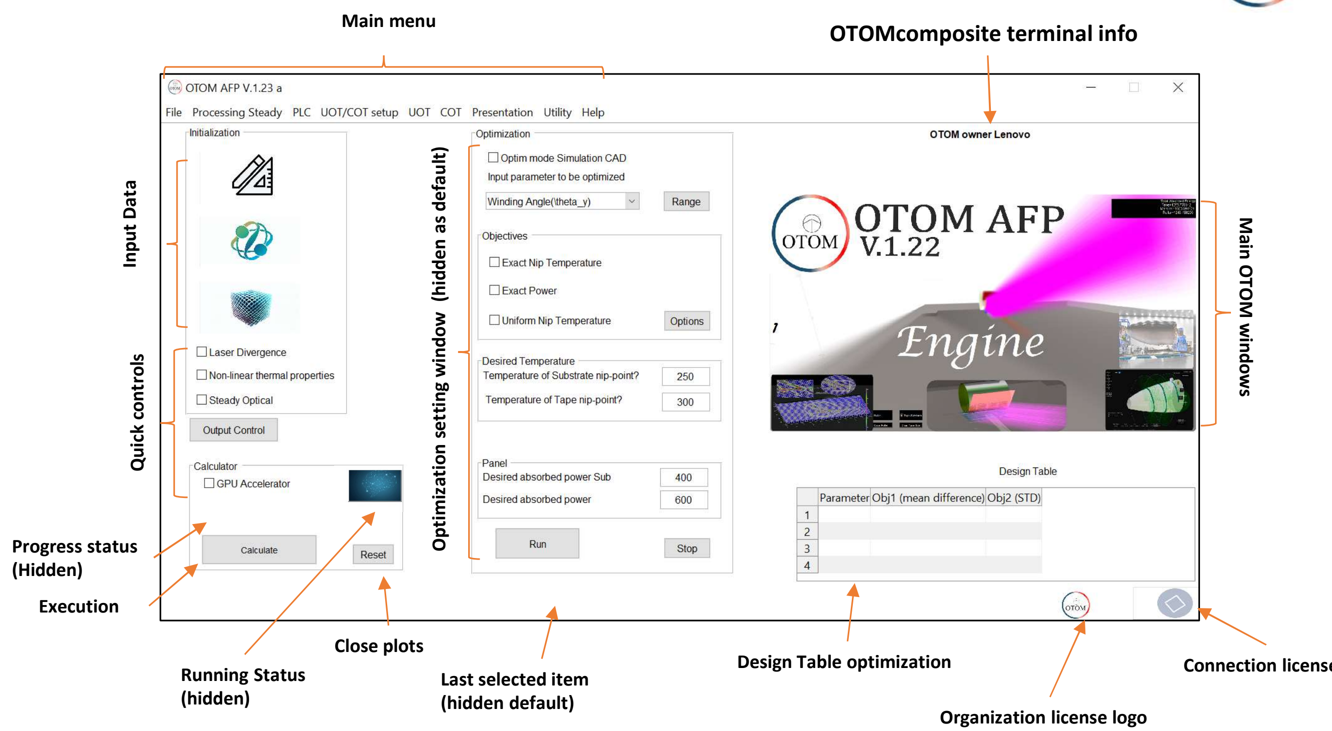
Outputs:

- **Ray-tracing outputs:** it includes coordinates of rays/reflections intersections and intensity distribution on the geometrical objects (Each kinematic step has a .txt file which includes the XYZ and absorbed power information).
- **Process temperature:** it includes temperature values at the laser-heated zone, consolidation zone, and global locations UOT (whole object).

Software Environment

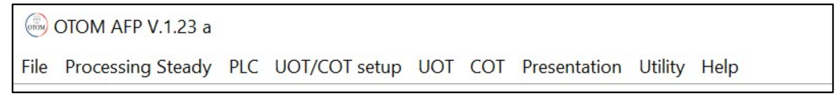
The start window of the AFP Engine software consists of several components:

- Main menu
- Input data
- Quick control
- Progress status
- Execution button
- Running status
- Close plots
- Last selected items
- Design table optimizations
- Organization license logo
- Connection license
- Main window
- OTOM composite terminal info

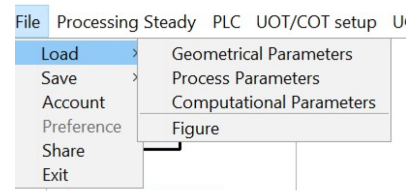


Confidential

Main menu: File



With the ***main menu***, you could select different functionalities. From the files, you could load and save different process parameters, geometrical parameters, and computational parameters. You could also load and save different figures, change your account name or organization, share your info for license registration and you could close the main window.



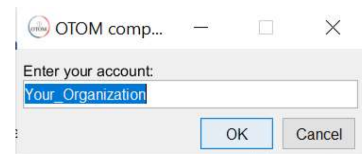
To **Load and Save input files** you could select from:

file> load > geometrical parameters, file> load > Process parameters, file> load > Computational parameters

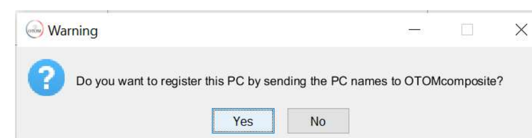
you could select the parameters that you previously used for your simulation

Further, you could similarly save the parameters from it, for example:

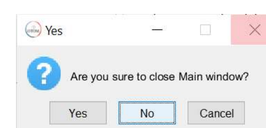
file> Save > geometrical parameters, file> Save > geometrical parameters, file> Save > Process parameters,



Account: If you select *File> Account,* from the main menu, you could enter your organization name. the file on the server of the OTOM composites will be checked and if your registration information is there, your license will be activated and the logo of your organization will be updated in the main window and connections with the OTOM composite will be established

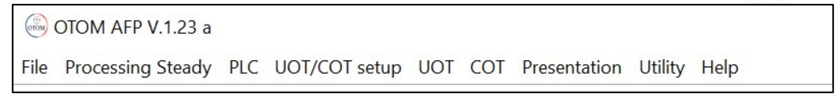


Share: To register your device information go to *Start> Share PC info*, your MAC address info will be sent to the OTOM server via your Email Microsoft Outlook application. You can check your Email to see the shared info. You must have Outlook installed on your device to use this functionality. (you might also need to close and open your Outlook if needed).

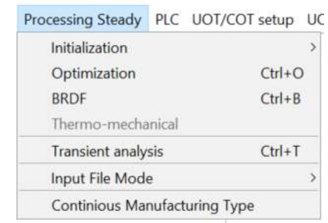


Exit: To exit the software you could go to **File > Exit** or you could directly close the window of the main software. A window will pop up and ask you if you really want to close the program.
All of the information during the running of the program will be saved in the OTOM.log which is in the directory of the installation.

Main menu: Processing Steady

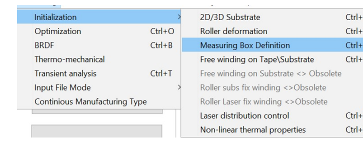


With **Processing Steady,** you can assign optical steady analysis settings from *Process study > initialization*, hide or show optimization, assign anisotropic reflection BRDF, perform transient thermal analysis, input file mode to bring new velocity and the laser power excel file and perform continuous manufacturing type for pressure vessel.

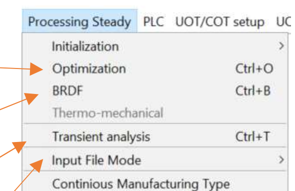


Optical steady analysis means that the optical phenomena remain the same but the thermal performance is changing. You could assign the setting via the *Processing Study > initialization.* For example, you could change the number of 3D points of the substrate or you could assign the Roller deformations, and also the measuring box definitions to compare the results with the thermal camera or you could assign the location of the laser spot and also the location of the laser source and the directions via *Free Winding on Tape/Substrate*. The data will be saved in the geometrical parameters.

You could also change laser distributions via the *Laser Distribution Control*, for example, you could change the size of the laser intensity of the laser and also laser divergence. You could also assign the *Nonlinear Thermal properties* for example if your specific heat capacitance will change with the temperature, or heat conductance will change with the temperature you could change those data in the Nonlinear Thermal Properties. You could also assign optical absorption mechanism when your material has a specific absorption for example if the surface has a nano-coating. All of these functionalities are available via *Processing Steady > initialization* from the main menu.



If you click on the Optimization button the optimization window will appear on the main window you could select the design variable and the limits of your optimizations. The default optimization window is designed for steady-state optical thermal simulations on cylinder or flat geometries, but if you select other functionality like an unsteady optical thermal (UOT) modeling this window will be adjusted accordingly.



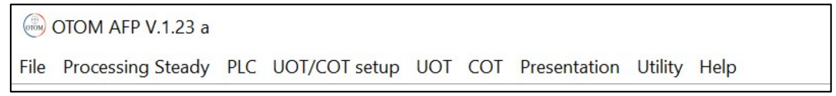
Anisotropic reflection behavior (BRDF) only works for the steady processing analysis you could assign different parameters to mimic the behavior of the fiber-dependent prepreg for example if you rotate the prepreg orientations of the fibers the reflection will be changed accordingly.

Transient analysis in the *Processing Steady* menu works based on steady optical analysis. You could assign the total time and initial conditions of the simulations and the simulation will be performed based on time steps and the total time. You could also save the video and animations of this simulation.

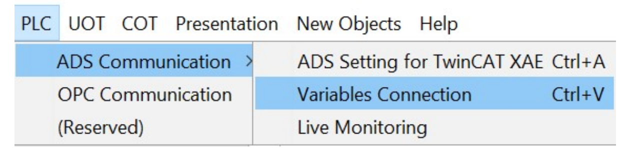
With the *input file mode* you could select the Excel file which includes all of the velocity and the laser power of the process. This means that the processing simulation will be a transient analysis which is based on steady optical modeling.

With the *Continuous manufacturing type,* you could perform simulations on the tubes and the pressure vessels which means that the condition of the gas inside and also the layers which will be placed near each other will be considered. This simulation is based on the steady optical and transient analysis.

Main menu: PLC

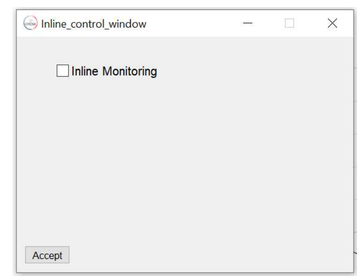


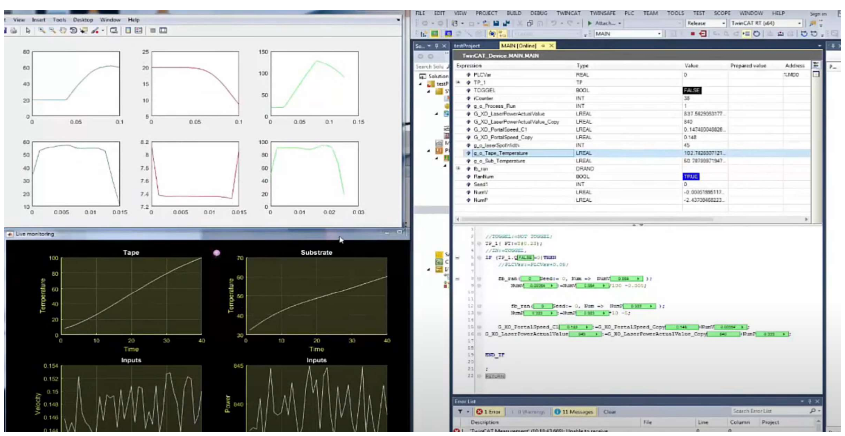
OTOM software can monitor and adjust the appropriate process conditions in a feedback loop control. The connection with ***PLC*** software like Twincat for transferring data via ***ADS communication*** as a secure and fast portal makes an in-line monitoring solution in a fraction of a millisecond.

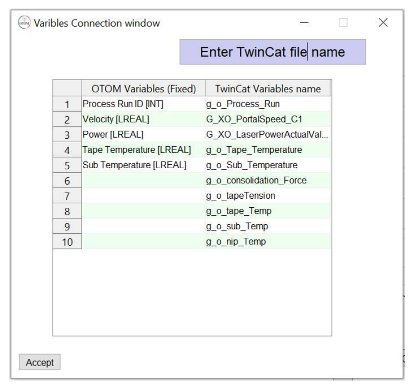


Plc functionality could be accessed from the main menu. It could be used for ADS communications. You could set up the ADS connection with the PLC software like a twin cat. You could assign the variables to indicate which variable from the OTOM software will be connected to the twin CAT software you see the window and you see the main file from the PLC. When the connection is established, you could transfer the data during the processing manufacturing and see the outcome via live monitoring. All data inputs and output between the two software will be plotted simultaneously in real-time.

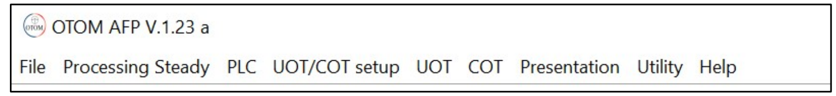
The other types of the connections like OPC could be requested via Customer demand.



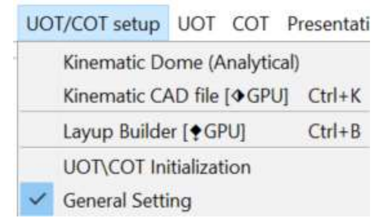




Main menu: UOT/COT Setup



UOT (unsteady optical thermal) model and COT (continuous optical thermal) analysis could be attained via the ***UOT/COT setup***. You could define the kinematics on the dome, pressure vessel, or kinematic on any CAD file. You could also define the layup via the layup builder. You could assign the initializations of boundary conditions for UOT&COT models. You could also change the general settings which connect the boundary conditions of the physical objects to the discretization of the numerical mathematical models. You could change all of these parameters via UOT/COT setup.



Via ***Kinematic Dome (Analytical)*** you could define the path on the cylinder and the domes for the analytical approach you could assign the direction of the laser and the movement of the roller on this geometry.

Via the ***Kinematic CAD File*** you could import the STL file from the CAD and you could assign the path on the CAD file, the discretization of the path, the direction of the laser, and the whole setup of the kinematics. this functionality works both on GPU and CPU but in case you have extra objects like a tape rack in front of the tape, it works only on the GPU

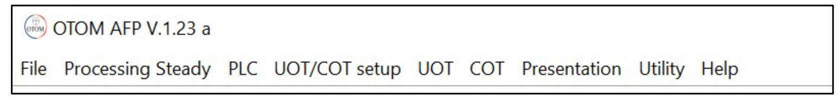
The functionality of the ***layup builder*** is only for the CAD file kinematics. When different kinematics are prepared via the *kinematic CAD file* you could assemble them via the *layup builder.* This works based on the CPU and the GPU. This means that you could set the order of the layup, the delay between the layup, and how many layups would be created. You could see the 3D layup representation. You could perform A simple analysis of the mechanical performance of the layup. The layup should be saved for the assigning boundary conditions in the next stage.

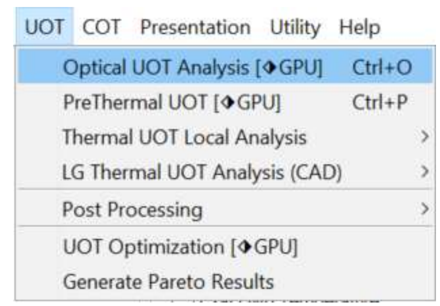
UOT\COT initialization is working for assigning the boundary conditions of the layers. For example, if the user like to change the thermal resistance between the layers and how are the boundary conditions with the below layers or upper layers, the user could assign them. You could assign the initials conditions of them.

Furthermore, the implicit and explicit solver of the finite element (global fem) could be defined here. You could change the thickness here in the case of the UOT analysis, and in COT analysis the thickness will be estimated automatically. Some settings regarding the curved geometry of the CAD file could be assigned here. These carved parameters help the software to understand and estimate what are the boundaries of each layer and based on that info they could build the layup and assign the boundary condition from node to node between the layers. This means that one node from surface A will be assigned to the specific node from surface B. This procedure continues for the whole node. Thus in case, there are the thousand of nodes per layer, all of these nodes will be assigned to each other accordingly

In The ***General UOT/COT Setting*** You Could Set The Temperature Range And The Intensity Flux Range Of The Analysis, You Could Set The Ambient Temperature, Air Convection Coefficient, Plied Tape Start Delay (When The Incoming Tape Will Be Placed On The Substrate) And Other Geometrical And Discretization Parameters Which Connect The Physical To The Computational Simulations Parameters. You Could Also Assign The Roller STL File Which Will Be Used In The Analysis Of The STL Cat Simulation.

Main menu: UOT





With the UOT tab you could set up the ***Optical UOT analysis***. It works based on the CPU and the GPU. After the optical analyses, the thermal point should be fed via the optical points. It will be done with the ***Prethermal UOT.*** It distributed the optical points to the thermal nodes. It also works based on the CPU and also GPU. The thermal UOT analysis works for the pressure vessel and flat panel based on the analytical approach. You could perform either a steady state analysis or transient analysis based on the analytical models.

For the cad thermal analysis, you should use the local global thermal UOT analysis. It also works based on the CPU and the GPU. You could then later perform the post-processing analysis and see the heat intensity distributions and the optical intensity distributions. It is in the post-processing tab of the thermal and optical analysis.

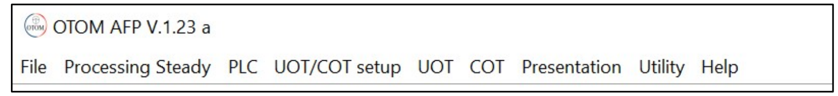
The optimizations of the UOT work are based on the both CPU and the GPU. For example, you could set the temperature at a fixed value during the roller movement on the carved locations so the laser power will be adjusted accordingly. This functionality works based on the analytical models.

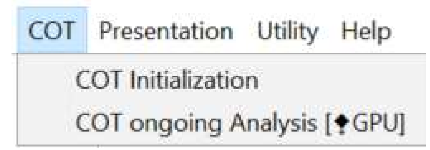
Finally, you could perform the Pareto analysis based on the outcome of the multi-objective optimization. To analyze this you need a set of the analysis which are obtained for the model. You could perform and demonstrate those designs to generate pareto results. Currently, it works based on UOT analytical models, and there are some limited functionalities for the UT CAD file which will be developed later in the future versions.

*** Please note that performing optical analysis and also kinematic for a large number of rays and path points is only efficient on the GPU. Thus, in the case of using the only CPU, it might take a longer time to analyze the kinematics and optical analysis, and Pre-thermal analysis.*

*** Please remember using a larger number of nodes and a larger number of rays requires a higher amount of VRAM. For the best performance experience is always recommended to use the GPU with at least 8 GB of Nvidia VRAM.*

Main menu: COT





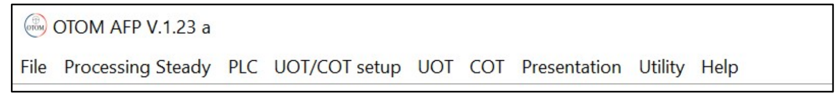
With the COT tab, you could perform continuous optical thermal simulations. It means that the optical analysis and the thermal analysis will be performed at each step. This would be very beneficial when you have a very complex layup. For example, if your layers are on top of each other in different directions. The software will estimate and understand which layup and which layers are on top, and they distribute the optical energy to the right and appropriate layers. Then the heat flux will be at the same time distributed and the thermal step will be performed. After one step again the same conditions and procedure will continue and all of the analysis of the optical data will be again performed. In other words, the data of the ray tracing will be saved on the text file.

This simulation only works based on the GPU to have the best experience possible for the users. You could assign and initialize your analysis via the *COT > COT initializations and* then select the COT ongoing analysis. The same procedure could be followed for the post-processing of the COT analysis. All of the data which will be generated via this analysis could be then later observed and demonstrated via the OTOM AFP viewer which is a supplementary software for the OTOM AFP analysis.

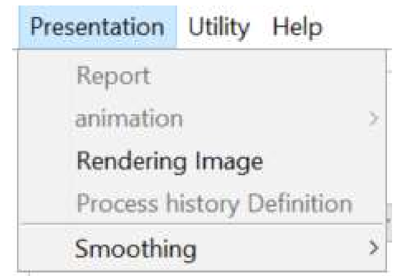
*** COT thermal models work based on the FEM global models which means that there would be no local or consolidation model. All of these steps will be done in the global finite element models.*

*** for the COT analysis guide, please refer to the Tutorial videos.*

Main menu: Presentation

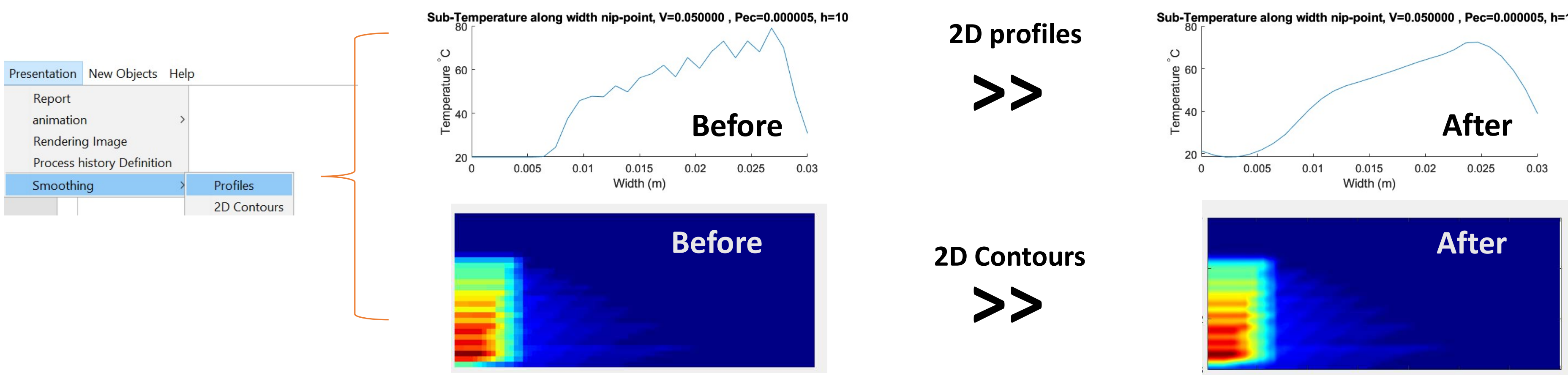


With the ***Presentation*** tab, you could present and save the images of the figures and also perform smoothing of the curves and the contours which will be used for the presentations or in the reports. Other functionalities are still under development and it could be added in the future release of the software.

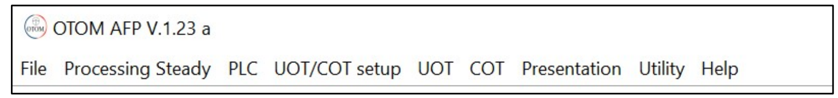


To save the image after all of the plotted images simply click on the **rendering image** *and* it saves all of the images of the current open plots and saves it on the folder directory installations of the software. It will be saved in ***\Results\Pics***

You could also select a ***smoothing*** function and select the specific figure which has 2D contour or line plots. You could assign the appropriate smoothing parameters to perform the smoothing of those plots. It makes line plots and colorful contours more continuous.



Main menu: Utility



With the ***utility*** tab from the main menu, you could define the different analytical mandrels. You could also convert the mesh file and change the sets.

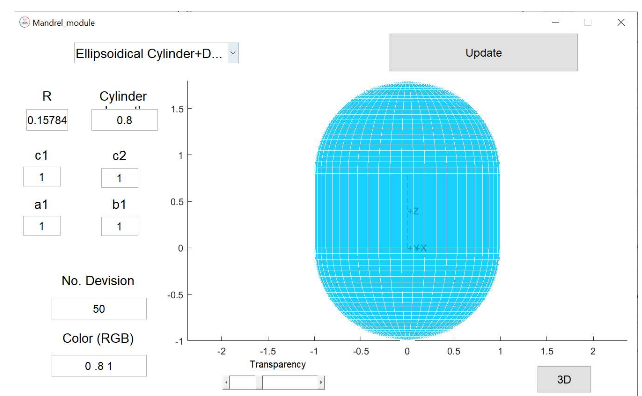
For example, for the analytical mandrel definitions you could change the size of the mandrel, the size of the vessels, the size of the domes on both sides, the number of divisions, and color transparency, and see the outcome geometry.

You could also import the mesh nodes and connectivity file and represent the mesh. You could then assign the sets for the mesh. For example, assign the upper surface or bottom surface. You could then save the mesh file in INP format to be used for the thermal analysis. Selecting nodes and searching could be based on the individuals or based on the tangent of the limitations of the surface. All of these options could be assigned by the user.

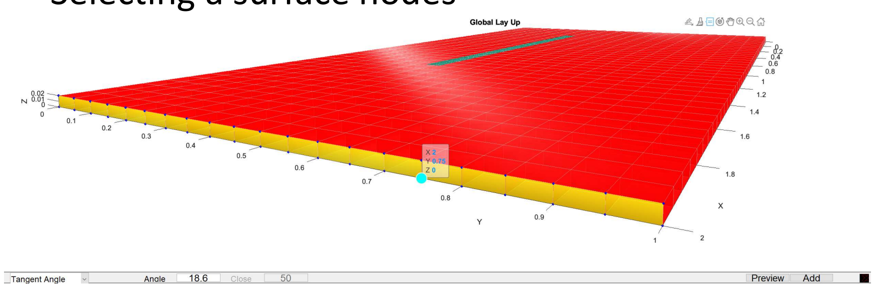
*** For the details of performing utility functions, please refer to the Tutorial videos.*

***Please note some of the functionality is not yet available and will be activated in the future release of the software.*

Mandrel Definition

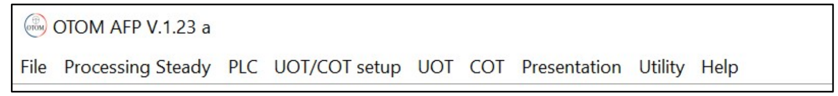


Selecting a surface nodes



Confidential

Main menu: Help



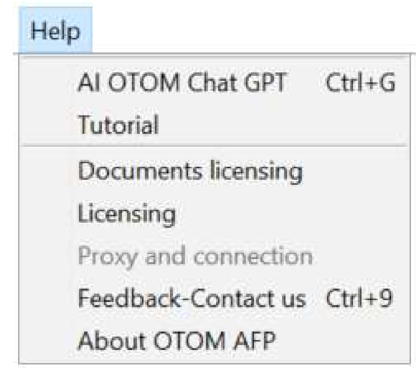
With the ***help*** tab from the main menu, you could call different functions. One of the very handy functions is the ***AI OTOM chat GPT*** you can ask questions and request a guide to navigate through the software. It’s an AI bot that helps you to decide and find the appropriate guides for the simulations (Your data will not be stored in OTOM software).

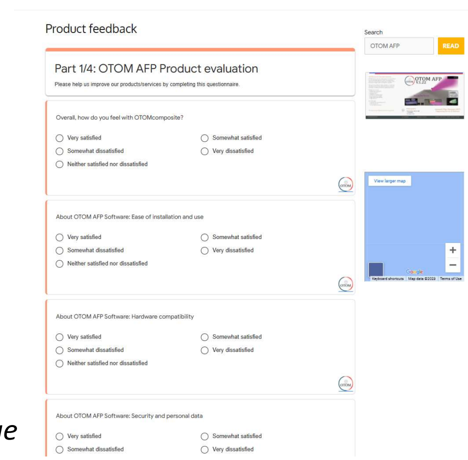
For performing tasks with the software, you could also select the ***Tutorial*** to see the documentation and the videos regarding using the software.

If you would like to change the ***Licensing*** or see how the licensing works you could click on the documents of the licensing.

In case you would like to give feedback or comments about the software or the service you could use ***Feedback contact us*** you will be directly connected to the website to write your feedback.

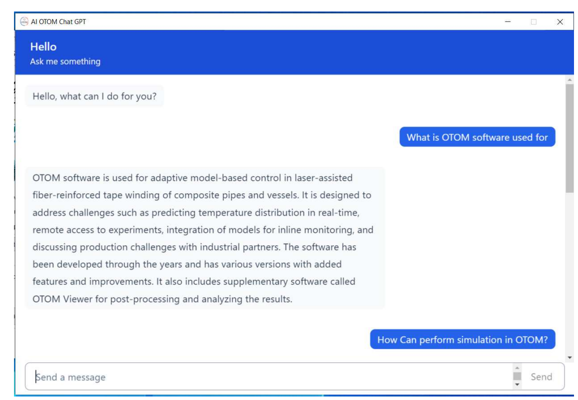
More information about the software or further details could be found via the ***About OTOM AFP.***



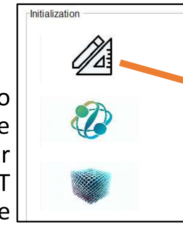


OTOM product feedback page

AI OTOM Chat GPT

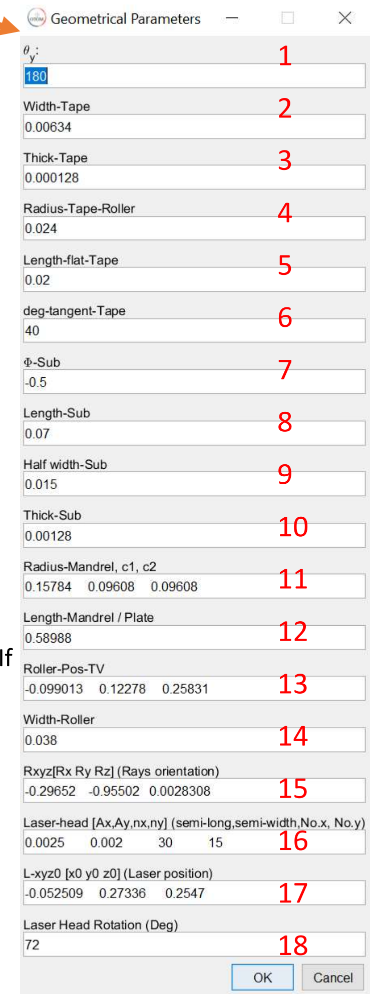


Input Data: Geometrical parameters

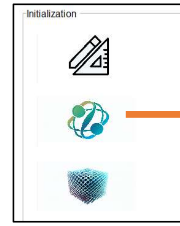


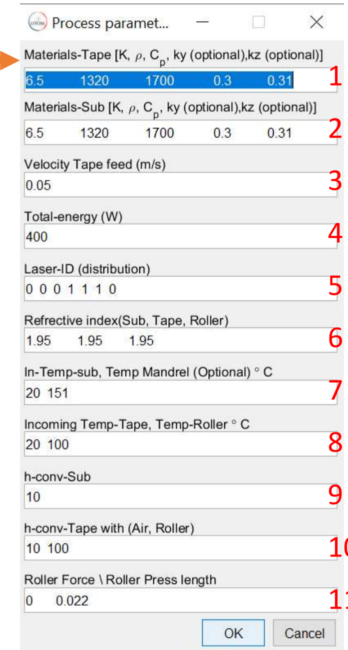
Geometrical parameters define the size of the objects in the AFP process setup. For example, it gives the directions of the roller, and also the sizes and thicknesses of the tape and substrate, the sizes of the vessels, the locations of the lasers, the orientations of the laser, the discretization of the lasers, and the number of the rays. It basically gives all of the geometrical parameters for a steady optical analysis. For the UOT or COT analyses some of the information will be used and some of them will be fed in the kinematic descriptions via the UOT/COT kinematic analytical or kinematic on the CAD file. Here we will have a look at each parameter and what are the definitions and what are the inputs for each variable.

1. The first variable is θy. It gives the orientation angle based on the degree of the roller. This value determines if the roller rotates around its main axis.
2. The width of the tape (Width-Tape) is the width of the incoming tape. This value might be changed in UOT kinematics.
3. The Thick-Tape is the thickness of the incoming tape. For the UOT/COT CAD analysis, this value is received from the mesh file.
4. Radius-Tape roller is the radius of the roller. This value will not be changed in the UOT CAD file. You should change it here.
5. Lenth-flat tape is the length of the tape after tangent positions with the roller. this will not be changed in UOT CAD file. It should be changed here.
6. Deg-tangent-Tape is a degree to which incoming tape is in tangent with the roller. This value could be changed only here.
7. Φ substrate is the orientation of the substrate with respect to the vessel in the analytical simulations. This will not be used in the UOT analysis.
8. Lenth-Sub is a length of substrate in the steady optical analysis. this will not be used in the UOT analysis.
9. Half-width substrate is a half-width of the substrate. This will not be used in the UOT kinematic analysis. You could change it here.
10. Thick-Sub is the thickness of the substrate for steady optical analysis. this will not be used for the CAD-based analysis.
11. Radius-Mandrel is the radius of the vessel and the other two parameters C1 and C2 are the radius for the domes. This is for UOT analytical analysis. If the first value is equal to zero then the mandrel is flat analytical then the C1 and C2 parameters are not in use.
12. Length-Mandrel/Plates give the dimensions to the size of the Vessel analytical or flat panel analytical.
13. Roller-Pos-TV (position translation vector) is the location of the roller in the optical steady analysis. this is not in use for the UOT analysis.
14. Width-roller is the width of the roller. this will be included in the UOT and COT analyses as well.
15. Ray orientations XYZ are the directions of the laser head. this will not be used for UOT or COT analysis.
16. Laser head dimensions (Ax, Ay) and number of divisions (nx,ny) input are the half size of the laser head and are in use in UOT&COT analyses as well.
17. Laser position L_xyz0 are the locations of the laser head. it is only for a steady optical analysis. This is not in use for UOT or COT analysis.
18. Laser head rotations are only for a steady optical analysis. This is not in use for UOT or COT analysis.



Input Data: Process parameters

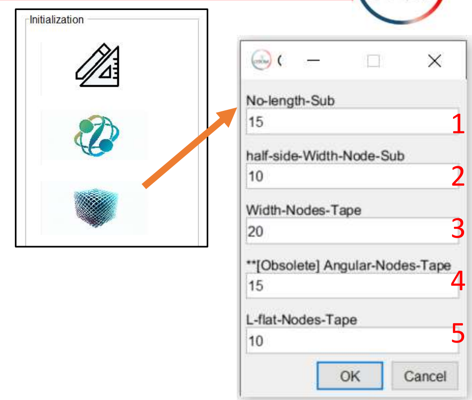




Process parameters define the material properties and the properties regarding the process like the total power of the laser or the velocity. you could also define the refractive indexes and boundary conditions of the tape or substrate, parameters regarding the roller force deformations in an optical steady analysis. some of these parameters will be also used for UOT or COT analysis. Herewith we will explain them further below.

1. The first process (Materials-Tape) row is the materials of the tape. It includes the X-axis conductivity coefficient, density, and the specific heat coefficient and also the conductivity coefficient along the y-axis and z-axis. The last two parameters are optional. In case you don't enter them here it is assumed their value is the same as conductivity along the X axis. Here y-axis is the fiber direction or lateral direction and the z-axis is the thickness direction. This will be used in the processing steady analysis and for the UOT analytical analysis. This parameter will not also be used for the UOT CAD analysis and COT CAD models. The material parameters will be gathered from the material properties from the UOT/COT initializations.
2. The Materials-Sub is the materials of the substrate and it has the same definition as Materials-Tape.
3. The Velocity-Tape is the Tape feed rate where the material is placed on the substrate. This parameter will also be used in the COT & UOT analysis.
4. The total laser power of each laser module will be inserted here. This parameter will also be used in the COT and UT analysis.
5. Laser distributions ID will be inserted here. You can choose from the uniform distribution, gaussian distributions, or advanced Gaussian distributions. You can find the related parameters ID via the ***Laser Distributions Control*** from the ***Processing steady*** > ***initializations.***
6. You can assign the refractive indices of the substrate, tape, and roller. These values will be used throughout the whole software in the COT & UOT analysis as well.
7. You can assign the incoming temperature (In-Temp-sub) of the substrate and the temperature of the mandrel. The second value is optional. These values will be only used for processing steady analysis. They will not be in use for the COT or UOT analysis.
8. You can assign the incoming temperature of the tape (In-Temp-Tape) and also the temperature of the roller here. These values will be also used in the COT&UOT analyses as well. In case the variable temperature will be selected in the UOT or COT analyses, these values will be ignored.
9. This is the convection of the substrate. This value will be only used for steady processing analysis. This will not be used in the COT or UOT analysis.
10. This value will determine the convection coefficients of the ape with the air and the roller before the consolidations. This will be used in the COT and UOT analysis as well.
11. This value is the Roller force and Roller press length. Both values will be used for the processing study analysis, and only the roller press length will be considered for the COT and UOT analysis. Roller deformations are only available in the steady processing analysis and not in the COT or UOT models.

Input Data: Computational parameters



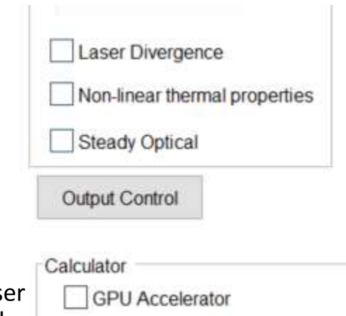
With the computational parameters, you can assign the discretization of the substrate and the tape. it is basically designed for steady processing analysis. only the tape discretization along the lanes and the width will be considered for the COT & UOT analysis. for the CAD-based analysis, the discretization along the path will be decided via the UOT kinematics.

***The computational parameters input window might be changed in the future release.*

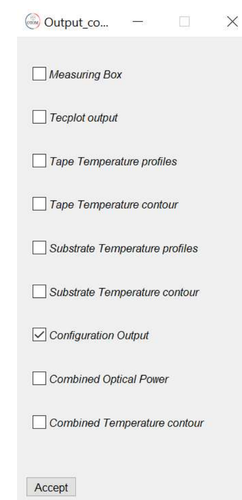
1. The first value is the amount of discretization along the length of the substrate. This will be only used for the study processing analysis. It is not in use for the COT or UOT analysis.
2. The half-side width of the substrate is the amount of discretization for the half-side of the substrate. It will be used for the study processing analysis. For the UOT or COT analysis, the number of divisions of the substrate could be changed via the UOT kinematics.
3. The number of the discretization of the tape weights for the incoming tape is assigned by this value. It will be used also for COT & UOT analysis for the applied tapes the values will be taken from the UT kinematics.
4. This value was obsolete.
5. The amount of discretization along the length of the tape will be here. This will be used for the incoming tape for the study processing analysis as well as the COT & UOT analysis.

Quick controls

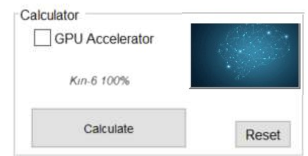
With the ***quick control's*** buttons, you could set ON and OFF the laser divergence, set On or Off the nonlinear thermal properties, and set On and Off the study optical analysis from the UOT simulations. You could also select the output controls where you control result windows of 2D line plots or contour plots to be represented. Finally, you could select the GPU accelerator. If you have an NVIDIA GPU on your device, you could use the functionality and the performance of your GPU which makes your simulations faster and easier for a certain number of simulations.



1. With the laser divergence thick box, you can turn on or off the laser divergence. The value of the laser divergence will be controlled via the laser distribution control. This function works for steady processing as well as UOT & COT analysis. This function for example for the UOT analysis should be changed before the optical simulations are effective.
2. The nonlinear thermal properties could be turned on or off. The values of the nonlinear properties could be changed via the ***processing steady > initializations*** > ***nonlinear thermal properties.*** This functionality is only available for steady processing. The linear thermal properties are not yet available for the COT OR UOT analysis.
3. The steady optical tick box is used for the UOT simulation. When the optical analysis through the steps remains almost the same, to save the computational cost you could select this item and then assign one step at the reference step. It tells the software that all the steps have similar optical inputs, so the software does not need to calculate optical inputs at every step.
4. The GPU accelerator tick box set the GPU to be used or not used when you have Nvidia GPU. It only works for certain functionalities in the COT & UOT analysis. The GPU accelerator does not work for steady processing analysis and analytical approaches. When you select the GPU accelerator the NVIDIA icon will be shown in the calculator frame in the main window. It also represents the amount of VRAM and the details of your GPU via the text box when you select this item.
5. With the output control button, you could select which window to be represented specifically for the processing and study analysis. For example, you could select the measuring box as the output or Tecplot output. The Tecplot output gives the PLT format to be represented in Tecplot software. The other figures which could be controlled here are the tape or substrate temperature profiles and contour plots. They provide the value along the nip point and along the movement direction of the roller. The configuration output is the setup of the whole geometry including the optical and thermal outputs. The combined optical power and combined temperature contours give the optical and temperature values on the carved tape and substrate. It was basically designed for the study processing analysis. For the UOT & COT analysis, you have extra plots for each object and each layer which are not represented here.



Calculator Frame



With the ***calculator frame*** box, you can start the simulation or stop the simulation. You can close all the windows with the reset button or you can start the calculations of the steady processing UOT optical UOT pre-thermal or COT. With these calculations, the window progress bar will be updated during the simulations. During the simulation also an animation will be represented on the right corner of the figure showing that the software is processing the numerical data. The progress after each layer and also the progress of the delays between the layers will be updated through the status bar. The progress bar is by default hidden as well as the progress animation icon. GPU accelerator icon is also here which is by default hidden.

To perform the simulation simply click on the calculate. The default calculation works based on the processing steady analysis. It only works when you approve the geometrical process and computational parameters. If you select the transient analyzers, it's also automatically going through transient calculations. If you select any functionality from the UOT like UOT optical analysis or UOT pre-thermal or UOT thermal analysis there is a string on the calculate will be changed based on the function that you selected. It calculates then the according functionality that you just select. If you hover the mouse over the calculate, an animation will be represented which shows the nature of this function for the UOT functionality and COT functionalities.

The progress of the calculation will be shown and when the analyzes are stopped the progress bar is 100%. During the simulation progress, you could select the reset to stop the analysis. You could also use the reset button to close all the open windows.

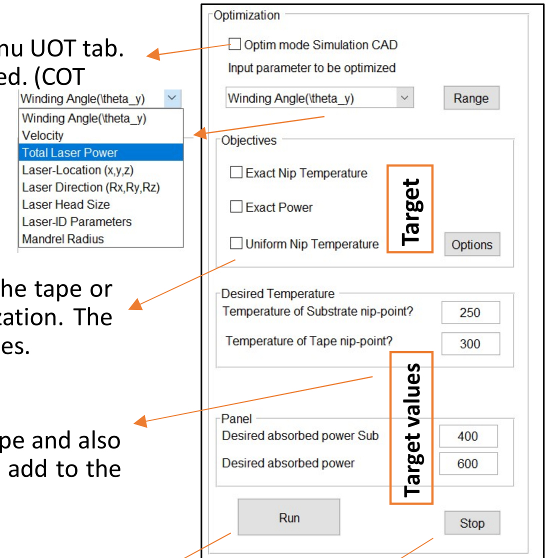
When you select an item for the UOT or cot analysis and a status bar below the calculator frame will be shown which shows the last selected item. This is by default hidden

Update Calculate text

Update GPU icon

Optimization Frame

You can access the optimization frame from the main menu or by pressing control + O on the keyboard. With the optimization frame, you could control the settings for the optimization, select the variables to be optimized, select the boundaries or limits select the targets for example target as the total laser power or the temperature of the nip point. Further, you could indicate if the optimization if it is for processing a steady analytical or the UOT simulation analysis.



By default, the optimization is for the processing steady analysis. For UOT analytical, you could select the UOT optimizations from the main menu UOT tab. For the UT CAD analysis, you could select ***Optim mode simulation CAD***. This will indicate that CAD simulation optimizations are to be performed. (COT optimization is not yet there).

Different variables could be selected for the optimizations, for example winding or placement angle, velocity, laser power, laser location, laser direction, laser head size, laser ID, and vessel size. You could also set the range of variables from the lower limit toward the upper limit.

Currently, three objectives for the optimizations could be selected. it could be the exact Nip-point temperature, or exact absorbed power on the tape or substrate, or the uniformity of nip-point temperatures. you could assign different options for that based on the multi-objective GA optimization. The related parameters to control the GA optimization could be changed via the options such as number of generations, population size, or tolerances.

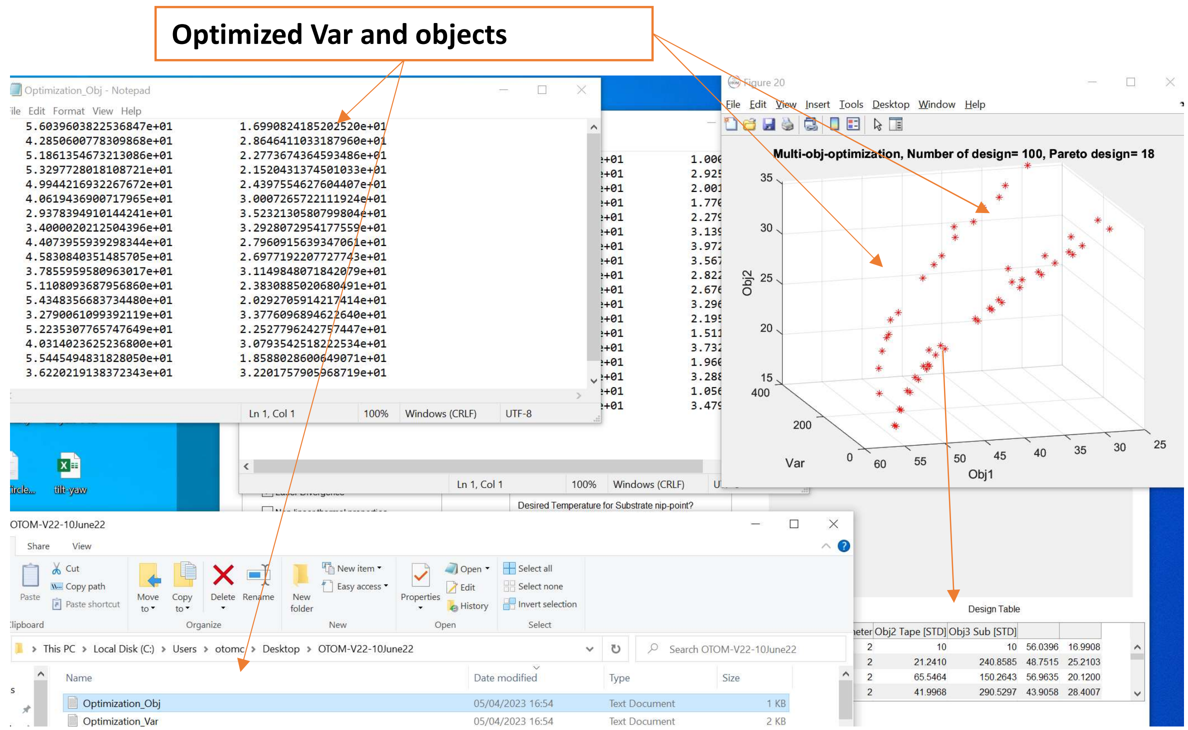
Based on the selected objective, the goal of the optimizations could be inserted. For example the desired temperatures for the substrate or tape and also the desired absorbed power on the substrate or tape. You could add that information here. The units are based on the information that you add to the system. Basically, it is SI, which means the degree of Celsius and also wat as for power.

When all settings are inserted for optimizations and all of the inputs are ready you could select ***Run*** to start the optimization process. During the optimizations design table will be updated and the figure of the main window will also be updated depending on the setting that you used and chosen for the optimizations.

During the optimization process, you could also select the ***stop*** to stop the whole optimization process. It breaks the simulation process.

Optimization Frame: Results

When your optimization is performed, you can find the optimization objective and variable results in the main folder and installation of the software. You can also see their representations of the optimization results in the figure graphically. All of this data could be represented either in the graphic in the design table of the main software and also on the text file. You see an example of every image of the multi-objective optimization with the number of the design of 100 and potato design of 18.



Info main window area

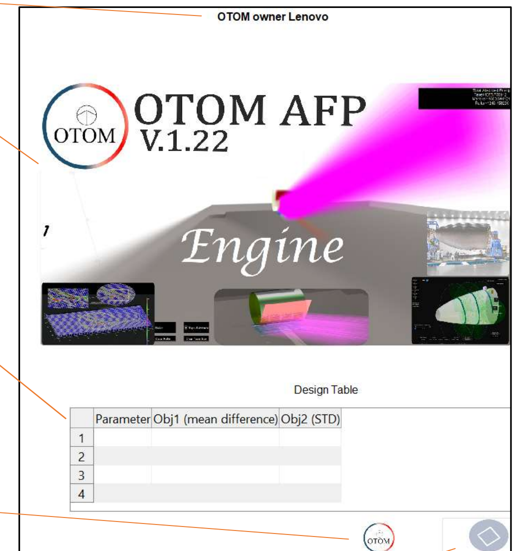
The right side of the software main figure gives information on the software licensing connected to the OTOM composite website. provide the design table information, and also the connection status with the license from the organizations that you inserted in the software.

On the top of the main figure image is the area of the terminal message from the OTOM composite license. The main info from your license will be indicated here, for example, if you have an annual license subscription for the software.

This is the main window of the software the picture will be updated every couple of minutes from the OTOM composite website. Some informative news from the bottom composite website regarding new products or new updates regarding the software will be indicated here.

This is the design table that is used for the optimization process during the optimization process. The data regarding the parameter's objectives will be updated.

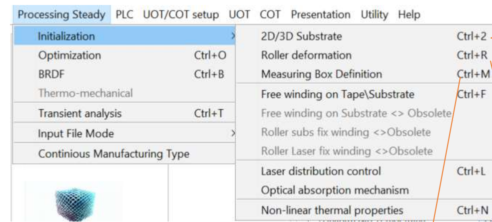
The logo of your organization related to your license will be shown here. If you change the licenses from different organizations the corresponding logo will be changed.

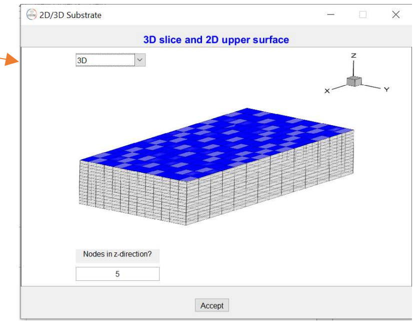


This is an animation showing the connection with the OTOM composite website. When it is enabled, it means that the connection is established for your license and when it is disabled it means that there is no connection available based on your license.

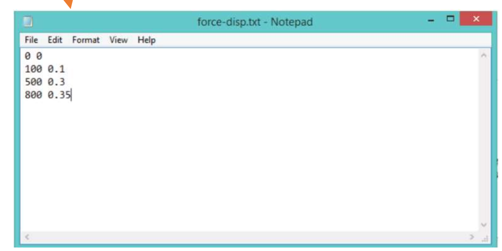
Processing initialization (1/3)

From processing a study and then the initialization tab you could reach the 2D/3D substrate. With this function, you could select the 3D finite difference of the substrate for the processing steady analysis. You could select the number of the nodes and you could also select if you would like to perform 2D or 3D analysis for the substrate. Please note that this is nothing to do with the UT analysis.

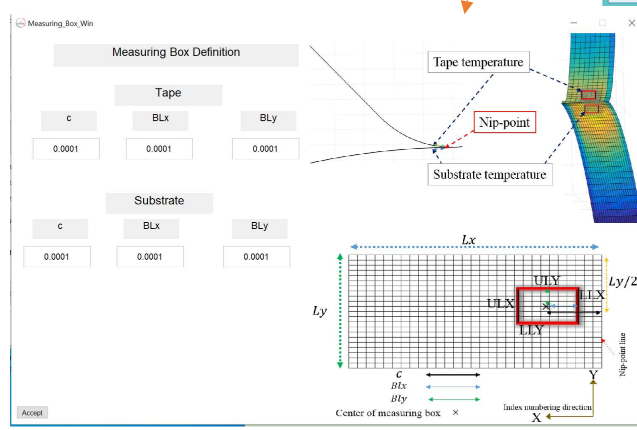


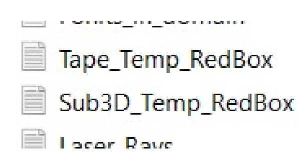


From ***Processing steady > Initializations > Roller deformations*** you could add the roller deformation text file. It should include the force and displacement data based on the force that you inserted in the process parameters. The software estimates the scale of deformations. It basically uses the curve fitting to find appropriate and corresponding deformation based on the force that you inserted. Please note that deformations here are only for processing steady analysis and it is not for UOT analysis. For UOT we only add the length of the roller press.



From ***Processing steady > Initializations > Measuring box definition*** you could access measuring box definition. This function is used to compare thermal camera information around the nip point area. You could define the Redbox area based on the distance from the side, from the nip point, and from the widths. In this area we'll get the values from the temperature nodes. You could then get the average values of the area based on definitions of the measuring box and compare them with the thermal camera results. Two text files will be created with the name ***tape temp Redbox*** and also ***sub3d temp Redbox.*** You could use these files' information for post-processing analysis. To enable this item you need to select control output from the main window of the software. This function is available for process study analysis this is not yet in use for UOT & COT thermal analysis.





Processing initialization (2/3): Laser Head orientation

From ***Processing steady > Initializations > Free winding on Tape\substrate*** you could access free winding on the tape and substrate.

1 With this window, you could select the target area and also laser location and orientations for the laser processing steady analysis. On the right side of the window, there are two buttons for removing the nip point and also adding the new point. When you add a new point, the roller position is set on the vessel.

2 On the left side, you will see the target you could select between the tape or substrate. If the target is on the tape, you could move the point on the tape and if the point is on the substrate, you could move it on the substrate.

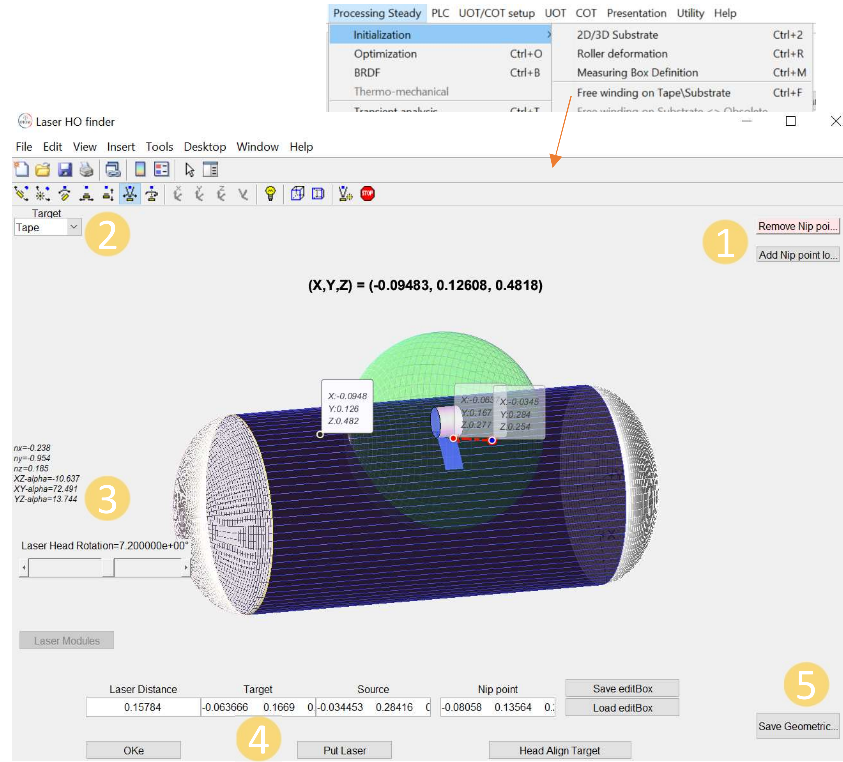
3 On the left side middle, you will see the orientation info of the laser with respect to the X, Y, and Z. Below that you could rotate the laser head by changing the slider. It will rotate the rectangular laser head.

4 At the bottom side, you will see several text boxes. You could see the laser distance, target, laser source, and nip point. When you select a point as a target and select a source as a laser source these text boxes will be updated. You could load and save these edit boxes to be used for later analysis. If you want to change the distance of the laser toward the nip point you could change the laser distance value. When you change this value, the green sphere size will be changed. This value is actually the radius of this green sphere. This green sphere helps you to put the laser source toward the target to approve the inserted value you could click on ***oke.***

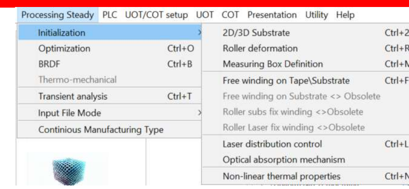
To assign the laser target you could click on ***Put laser***. It will create the laser direction from the laser source toward the laser target either on the tape or on the substrate. When you move the laser target, the green sphere will not be moved automatically. You need to change it manually via the ***Head align target***. If you press this button, the center of the green sphere will be moved toward the point after the target.

5 When you are finished with assigning the laser location and the laser target and orientations of the laser you could select ***save geometrical parameters***. The values in the geometrical parameters will be updated accordingly.

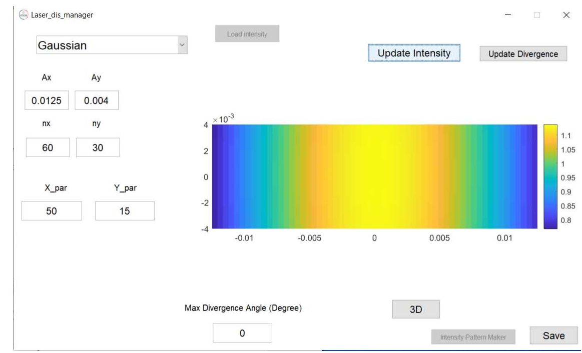
This function was designed for processing a steady analysis. This means that this does not have any influence on the UOT or COT analysis.



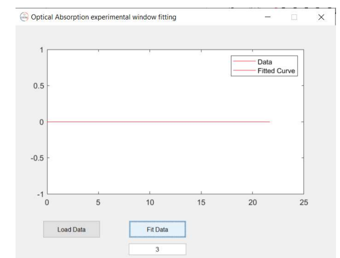
Processing initialization (3/3)



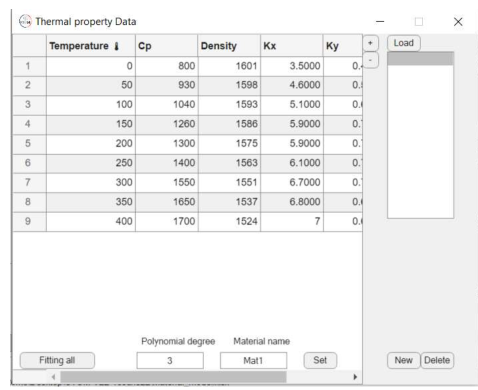
From ***Processing steady > Initializations > Laser distribution control*** you can find laser distribution control. This function helps you to design and determine the appropriate laser ID parameters. You could select the uniform type, gaussian type, or advanced Gaussian 4D type. You can assign the size of the laser head via the AX and AY. You can also assign the discretization along the X&Y axis by nx, ny. The parameters regarding the Gaussian parameters or Gaussian 4D are the ***x-par*** and ***y-par*** parameters on the left side of the figure. At the bottom, you could change the maximum divergence angle. This value is an angle at the corner of the rectangular laser head. The divergence value of the rest of the points on the rectangle will be linearly changed from the center of the rectangular laser head. The software calculates the divergence of each point accordingly. You could click the ***update intensity*** button and ***update divergence*** button to update intensity contour and divergence ray representations on the main figure. You could also represent the laser distributions in 3D by clicking on the ***3D*** button. When you finished the assigning parameters of the laser distribution control you could save the data by clicking on the save button. All the parameters regarding the laser distributions will be updated in the process parameters data. To use the new laser distribution ID you should open and accept process parameters.
**Importing load intensity from the experimental observation is also possible which will be released in future versions.

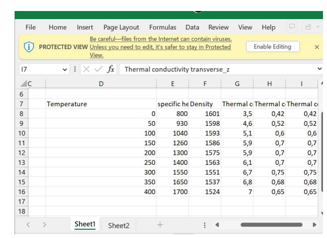


From ***Processing steady > Initializations > Optical absorption mechanism*** you can find optical absorption experimental window fitting. This function helps you to define new absorption mechanisms based on the experimental data. For example, if you have a nano-coating or a different type of surface where the absorption mechanism does not follow the Fresnel equation. This function is still under development and will be fully released in the future versions.



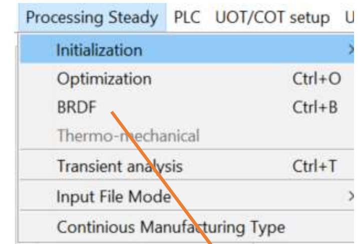
From ***Processing steady > Initializations > Thermal property data*** you can find nonlinear thermal properties definition. You can import the Excel file of the temperature-dependent material property like a specific heat density and thermal conductance. You can do this by clicking on ***load*** button. You could then modify the data, set it to the polynomial degree for the fitting, set the material name, and perform the fitting of the data. You can perform the fitting by clicking on ***fitting all*** button. Therefore, the software uses the fitting data to estimate the value of the thermal properties at each temperature. This functionality is currently only available for processing a study analysis it is not yet available for UOT thermal analysis or COT thermal analysis.

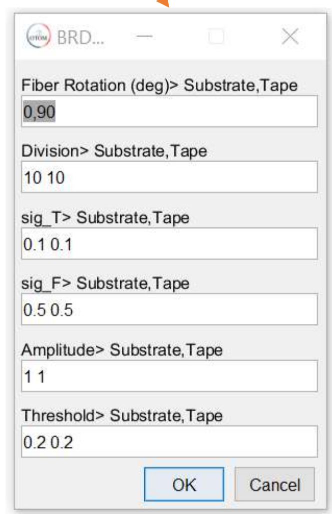




BRDF (anisotropic reflection)

From ***Processing steady > Initializations > BRDF*** you can select BRDF anisotropic reflection model. This model could be used for a steady processing analysis. This model is not yet implemented for UOT or COT optical analysis because of the computational expenses. The BRDF model only works on the CPU currently.

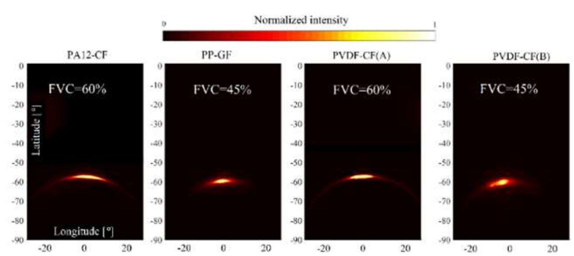


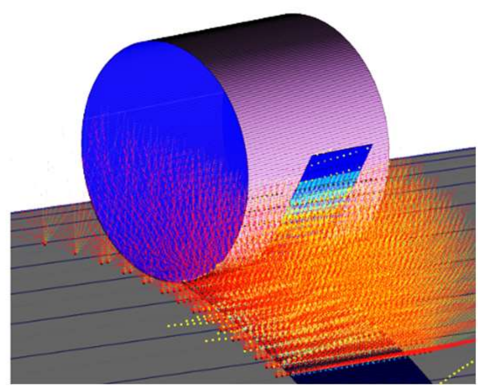


With the RDF optical analysis, you could define the fiber directions. You could also define the density of the fibers and fiber diameter and fiber sizes with the BRDF parameters.

1. The first value in the text box is the fiber rotations for the substrate and for the tape based on the degree.
2. The next parameter is the divisions for the substrate and tape. It is basically the number of rays per direction. The higher number brings more computational time.
3. The next two parameters Sigma T and Sigma F are the BRDF parameters. For details of the BRDF equations please check the related documentation of the software.
4. The amplitude of the substrate and tape is the maximum normalized value for the anisotropic reflection. It is recommended to not change this value.
5. The last parameter is a threshold for the substrate and tape. This value determines if the associated anisotropic energy is below threshold, the ray is not considered for the ray tracing analysis. If you put the threshold lower more anisotropic rates will be considered. The BRDF reflection functions decide the energy value for each ray.

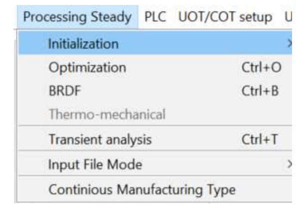
Please check the below reference for formula descriptions of the BRDF and implementations in OTOM: https://www.sciencedirect.com/science/article/pii/S1359835X21001263



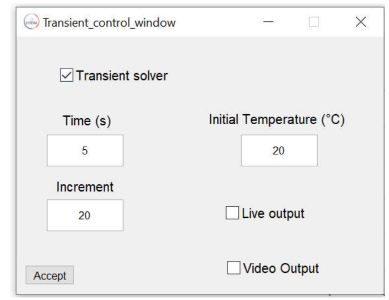


Confidential

Processing steady: Transient thermal Analytical & Input file mode

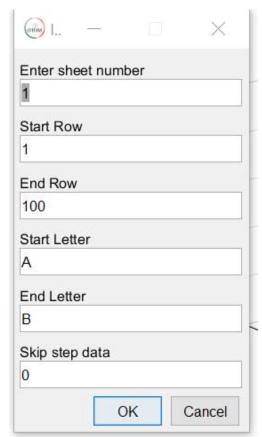


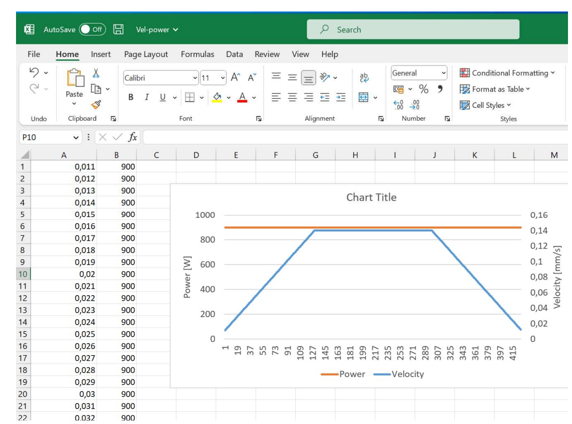
From ***Processing steady > Initializations > Transient Analysis*** you can select transient analysis. This analysis is for processing a steady processing analysis. You can here select the transient solver, the total amount of simulations, the initial temperature, and the increment, select the live output, and if you want to save the animations of the simulation select the video output. When you set all of the settings, you could click on ***accept***. Please note that this function is not related to the UOT or COT analysis.



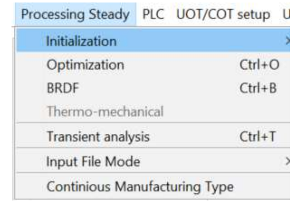
** UOT or COT analysis is by nature transient, and you select all of the boundary conditions and initial conditions we are the UOT & COT initializations.
** Please also note that when you use input file mode the increment will be adjusted based on the amount of data that you fit into the software. Thus, only the total time will be considered.

From ***Processing steady > Initializations > Input File mode > read file*** you can select the Excel file containing the velocity and the laser power. You could use this information for performing transient processing and a steady analysis. This model is not related to the UOT or COT analysis.
Here you can select the sheet number of the Excel file, starting row and the end row, and the corresponding letter column here you will see them as A&B, and finally you could select a ***skip step data***. The skip step data will skip the data between two consecutive values. For example, if you have data from one to 100 and your skip step data is 10, they will read the data every 10 steps. For example, it goes as one 11, 21, 31, etc.
This will be useful for large amounts of data for example when your measurements are recording the data at very small time steps but you need to use fewer data as possible to save computational cost so you read only a smaller data and still, you get your required output.

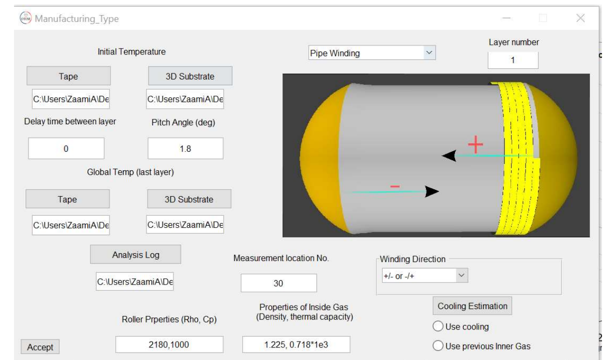




Continuous manufacturing type



From ***Processing steady > Initializations > continuous manufacturing type*** you can select continuous manufacturing type to perform processing a study analysis on the vessels and tubes. This model works based on the transient processing study analysis. This is not related to the UOT or COT analysis. Here you can select the pipe winding which is back and forth movement on the vessels and tubes. For detail modeling information please check the published paper below: https://journals.sagepub.com/doi/abs/10.1177/0021998320944598



With this feature, the simple movement on a vessel or tube will be simulated. The movement can be +/+ or +/- representing back and forth, or in one direction. The initial conditions of previous layers could be then assigned. This method works as a simulation of one round, then data will be saved and again new setting should be assigned by the user. The cooling estimation inside the tube and vessel is estimated using the thermodynamic formula. After assigning the setting the simulation will be ready by pressing calculated.

Please note the data from previous layers should be saved and loaded to have continuous manufacturing.

On the top right side, you could select the pipe winding and also select the ***layer number***. The layer number gives the thickness of the substrate. Currently, only the pipe binding is available.

From the left side, you could select the tape and substrate's ***initial temperature conditions***. You could import them from the previous analysis. You could find them in the directory after the installation folder. If you leave them empty the initial condition is considered as the ambient temperature which is 20 degrees Celsius.

The ***delay time*** between the layers is based on the seconds. If you put a number, there would be a delay at the end of the placement to the next layer. Then the material will be cooled, and the condition of the gas inside would also be changed.

The ***pitch angle*** defined the movement of the roller from one side toward another side. It is somehow the placement or winding angle. This value is not yet in use. Thus currently the pitch angle will be decided based on the length after the vessel, the time of the simulations, and the amount of discretization.

The global temperature is the last condition of the playset substrate and tapes. You can find the information from the last analysis in the directory installations this file is called the global temperature which was saved as text file. If you leave them empty the last step temperature will be considered as an ambient value which is 20 degrees Celsius.

Analysis looks save all of the information regarding the locations where the information was saved it should not be changed from one layer to another layer. When you simulate one layer, the analysis log will be updated, and you could select the same file for the next layer.

Measurement location number is discretization along the length of the vessels or tubes. The software recorded data on these points and the calculations for the estimation of the cooling of the layers and the conditions of the gas inside the vessel or tubes will be calculated on these points.

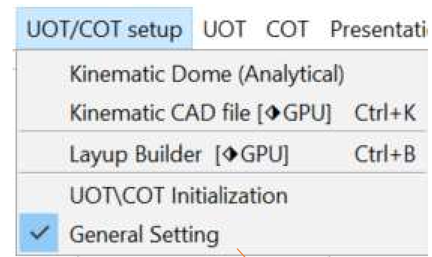
With winding direction, you could select the movement kinematics if you go back and forth or only along One Direction.

You can insert the roller properties and also the properties of the gas inside via the below two text box areas. For the roller, you get the density and specific heat to estimate the temperature. A similar parameter could be inserted for the gas inside the vessel or the tubes.

After performing a layer simulation, you could select cooling estimations. An analysis will be performed on the measurement locations to estimate heat transfer between the tape and the substrate as well as the gas inside the tubes or vessels. You could either use the previous in your gas or use the cooling information for the next layers you could select them by clicking on the radio buttons use cooling and use previews in your gas. When you inserted all of the settings you could click on accept to save the data and to be ready for transient processing steady analysis.

UOT/COT setup: General Setting

From ***UOT/COT setup*** you can set up and initialize the kinematics for UOT & COT analysis build the layup, assign boundary conditions for the UOT & COT analysis, and set the general setting for those analyses. Some function works on the CPU and some function on the GPU Please note that the performance on the GPU is usually faster than the performance on the CPU.



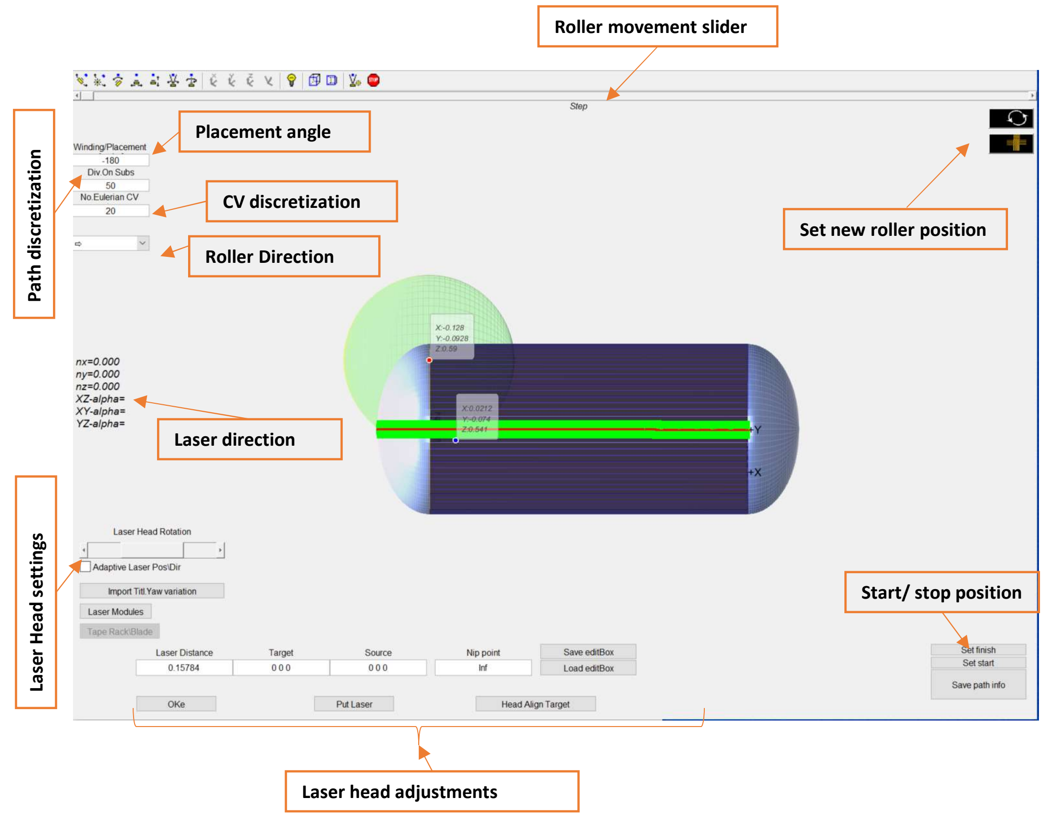
The general UOT and COT settings will be used in the UOT&COT thermal analysis.

- The first item is a temperature range Clim. It indicates the lower limits and upper limits to represent the temperature contours.
- the second item is the intensity flux range. It indicates the lower limit and higher limits for representations of the absorption power.
- The temperature ambient is there ambient temperature that is used in the COT&UOT analysis. If other values in UOT initializations are used then this value will be ignored.
- The air convection coefficient is used in the UOT&COT analysis for the convection coefficient. This means that the value here is considered for your COT&UOT analysis rather than the value in process parameters.
- ***"Plied tape start"*** is the delay when the incoming tape will be placed on the tooling. The dimension is second [sec] here. It means that after a certain amount of time the placement under tooling we'll start. Until that time there wouldn't be any new elements of the plied tape under the substrate at the beginning of a placement after layer.
- ***Control volume mesh multiplier*** and ***Grid refiner*** is the parameters that are used to connect physical properties to the discretization numerical properties of the simulation. You do not need to change these values. In case of a complex curvature situation, this value might be changed to give a better representation and calculation of the layup. With this value, the layup boundary conditions between nodes of each layer will be constructed.
- Vicinity ratio is the relative ratio around the point. This will be used to eliminate other points when the software search for neighboring points. This helped the software to limit the search scope and improve performance. For most analyses, you do not need to change this value.
- ***Roller shift margin factor*** and ***roller clearance factor*** are numbers that are multiplied by roller radius and tape thickness to give the clearance between the incoming tape positions of the tooling and the distance with the tape rack if applicable. It is recommended not to change this value till when you have a different type of roller geometry or roller deformation.
- ***"Roller STL address"*** is the roller STL file for consideration for UOT&COT analysis. The software uses this STL file and put it in the optical analysis. Please note that the roller dimension will be adjusted according to the geometrical parameters. You don't need to worry about the dimension change, the software can adjust the dimension in three axis of the XYZ accordingly based on the geometrical parameters from the main window.

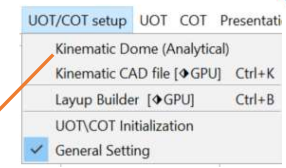
UOT Kinematic Dome (Analytical)

You can see here the high-level representations of the kinematic Dome analytical UOT. The window consists of several main elements:

- 3d representations of the vessel and domes
- Set new roller positions
- Placement angle
- Control volume discretization
- Roller direction
- Path discretization
- Roller movement slider
- Laser directions
- Laser head rotation
- Adaptive laser head position tick box
- Yaw and tilt variation import
- Laser modules definition
- Laser distance value
- Target position value
- Source position value
- Nip point position value
- Load the edit box and save the edit box
- OK to accept laser distance and auxiliary green sphere
- Put a laser to set the direction of the laser
- Head align target to set the auxiliary sphere toward the center of the target
- Set the start and set finish positions for the roller placement
- Save path info to store all of the information regarding kinematic



UOT Kinematic Dome (Analytical)



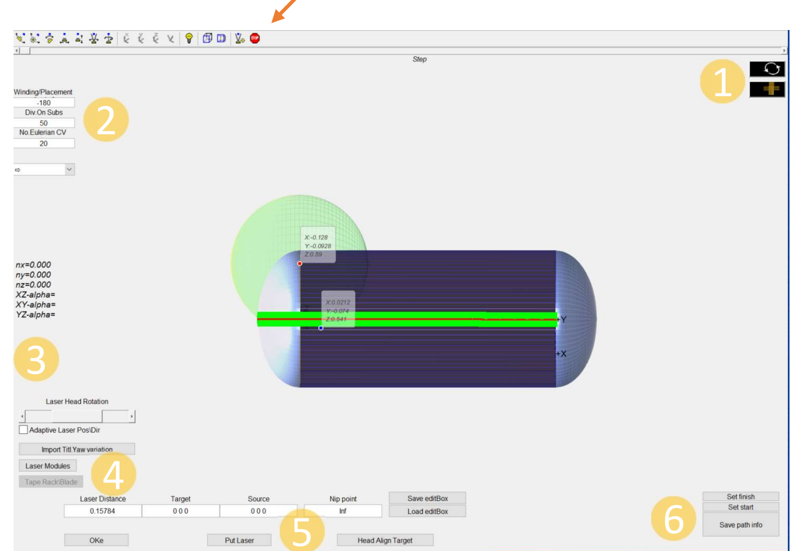
1 With these two buttons ***changing the point of the path*** and ***adding a new path***, you could select the target area for the path for UOT analytical analysis. The first one is for removing the nip point and second for adding the new point.

2 With these text boxes you can change the winding or placement angles, the number of the division on the substrate along the path, and also the number of control volume points. You can also change direction which will be ***forward*** or ***backward*** via this arrow direction.

3 On the left side middle, you will see the orientation info of the laser with respect to the X, Y, and Z. Below that you could rotate the laser head by changing the slider. It will rotate the rectangular laser head. When you click on adaptive laser position or directions checkbox the orientations and locations of the laser head will be moved automatically by changing the positions of the roller.

4 Another function is related to the ***laser head tilt and Yaw orientations*** and also the number of laser modules. You can change them by clicking on import tilt your variation. You should then open a Excel file that includes tilt value or tilt and your values. They should be column vectors that match the number of steps of the roller movement. These tilt and Yaw values are their relative variations of the orientation of the laser head toward the target point. With the ***laser modules***, you could add different laser heads for the complex setups as the grids. You can set up the number of laser modules in two directions and the distance between them

5 At the bottom side, you will see several text boxes. You could see the laser distance, target, laser source, and nip point. When you select a point as a target and select a source as a laser source these text boxes will be updated. You could load and save these edit boxes to be used for later analysis.



If you want to change the distance of the laser toward the nip point you could change the laser distance value. When you change this value, the green sphere size will be changed. This value is actually the radius of this green sphere. This green sphere helps you to put the laser source toward the target to approve the inserted value you could click on ***oke.***

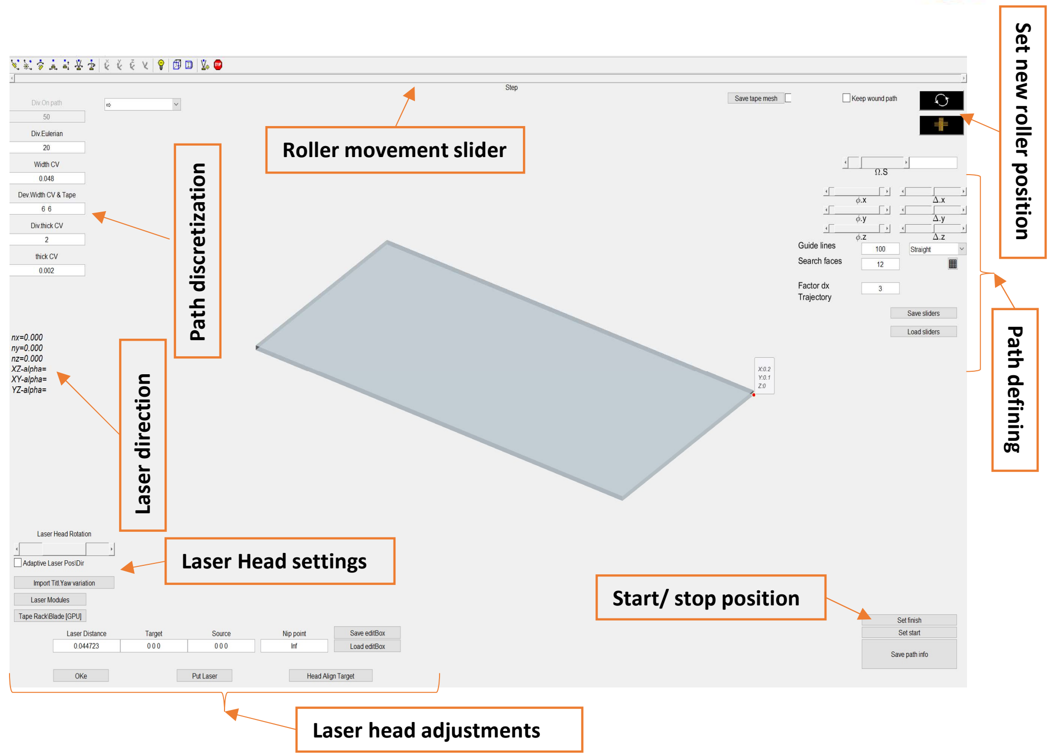
To assign the laser target you could click on ***Put laser***. It will create the laser direction from the laser source toward the laser target either on the tape or on the substrate. When you move the laser target, the green sphere will not be moved automatically. You need to change it manually via the ***Head align target***. If you press this button, the center of the green sphere will be moved toward the point after the target.

6 When you are finished with assigning kinematics, you could select ***set start*** and ***set finish*** to indicate at which step the start is and at which steps it will be finished. Then you can select ***save path info***. The kinematic UOT file will be created and all information regarding the laser locations, orientation and discretization will be saved on that file.

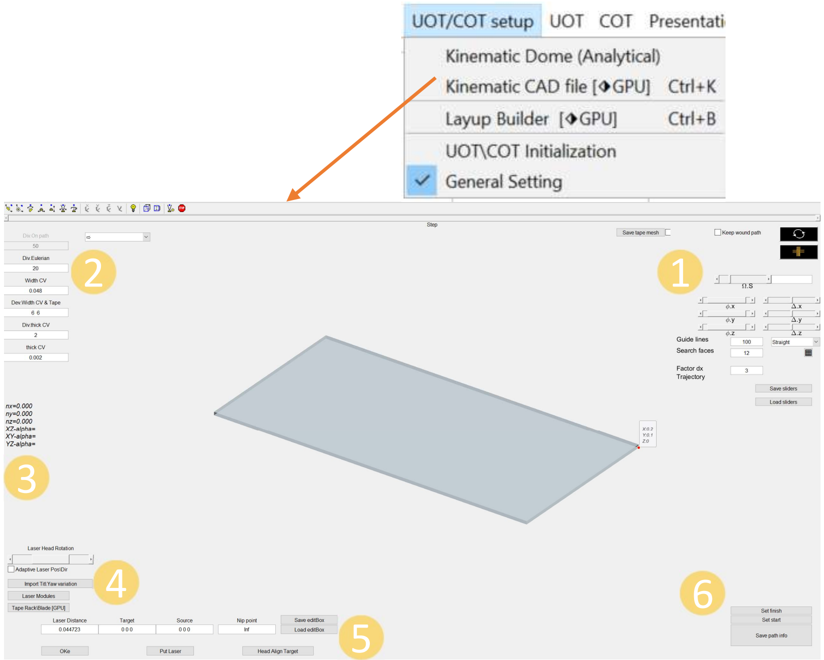
UOT kinematics CAD model

You can see here the high-level representations of the kinematic CAD UOT. This function can also be used for COT analysis. The window consists of several main elements:

- 3D representations of the CAD model
- Set new roller positions
- Path definition sliders and orientation definition
- Load and save path definition information
- Control volume discretization
- Roller direction
- Path discretization
- Roller movement slider
- Laser directions
- Laser head rotation
- Adaptive laser head position tick box
- Yaw and tilt variation import
- Laser modules definition
- Laser distance value
- Target position value
- Source position value
- Nip point position value
- Load the edit box and save the edit box
- OK to accept laser distance and auxiliary green sphere
- Put a laser to set the direction of the laser
- Head align target to set the auxiliary sphere toward the center of the target
- Set the start and set finish positions for the roller placement
- Save path info to store all of the information regarding kinematic



UOT Kinematic CAD model



1 With buttons ***changing the point of the path*** and ***adding a new path***, you could select the target area for the path for UOT CAD analysis. The first one is for removing the target and the second is for adding a new point. You could change the size of the plane for path definition or orientations. This will be explained in detail in the next section. You could also save the ***INP tape mesh*** after creating the pass to be used for thermal analysis.

2 With these text boxes you can change the number of the division along the path, and also the number of control volume points. You can also change direction which will be ***forward*** or ***backward*** via this arrow direction. This will be explained in detail in the next section.

3 On the left side middle, you will see the orientation info of the laser with respect to the X, Y, and Z. Below that you could rotate the laser head by changing the slider. It will rotate the rectangular laser head. When you click on adaptive laser position or directions checkbox the orientations and locations of the laser head will be moved automatically by changing the positions of the roller.

4 Another function is related to the ***laser head tilt and Yaw orientations*** and also the number of laser modules. You can change them by clicking on import tilt your variation. You should then open a Excel file that includes tilt value or tilt and your values. They should be column vectors that match the number of steps of the roller movement. These tilt and Yaw values are their relative variations of the orientation of the laser head toward the target point. With the ***laser modules***, you could add different laser heads for the complex setups as the grids. You can set up the number of laser modules in two directions and the distance between them.

You could also import a tape rack blade. This object is supposed to be in front of the incoming tape and it is only available for UOT CAD analysis based it on the GPU.

5 At the bottom side, you will see several text boxes. You could see the laser distance, target, laser source, and nip point. When you select a point as a target and select a source as a laser source these text boxes will be updated. You could load and save these edit boxes to be used for later analysis.

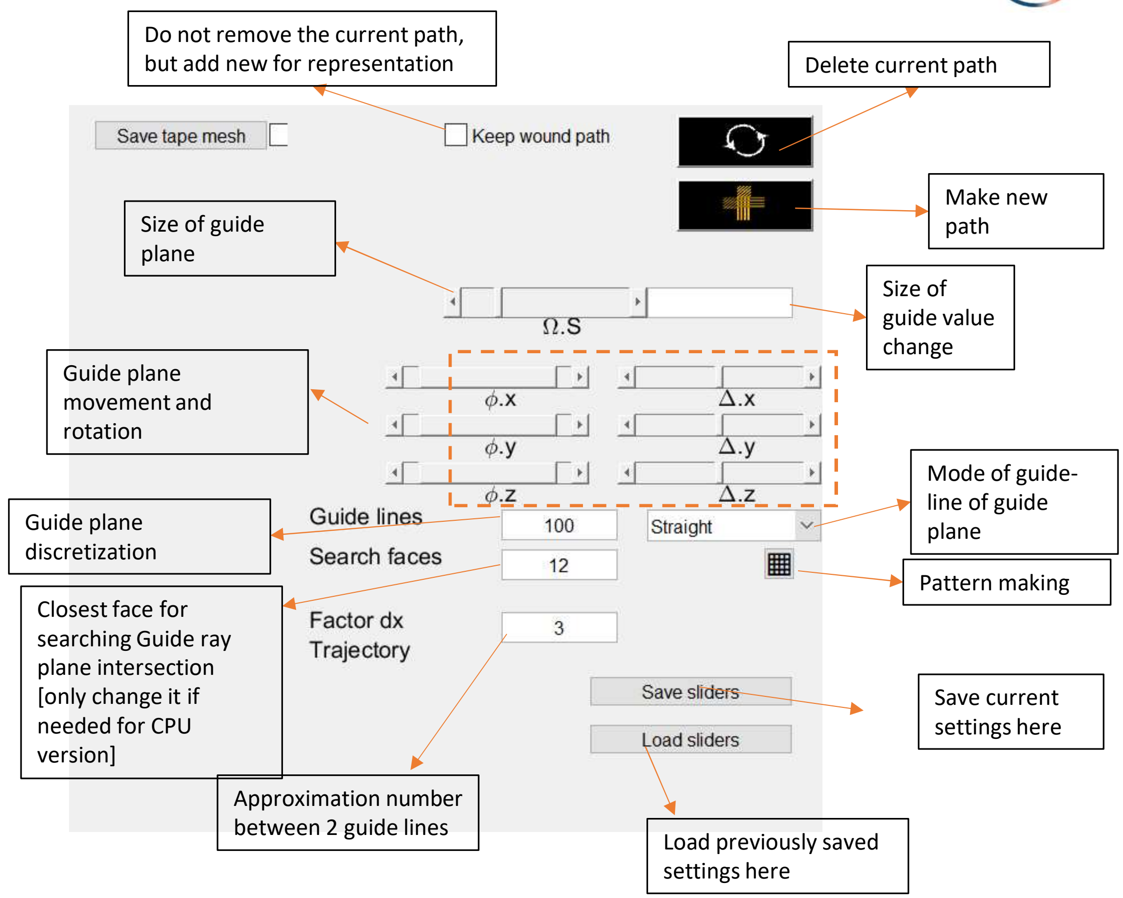
If you want to change the distance of the laser toward the nip point you could change the laser distance value. When you change this value, the green sphere size will be changed. This value is actually the radius of this green sphere. This green sphere helps you to put the laser source toward the target to approve the inserted value you could click on ***oke.***
To assign the laser target you could click on ***Put laser***. It will create the laser direction from the laser source toward the laser target either on the tape or on the substrate. When you move the laser target, the green sphere will not be moved automatically. You need to change it manually via the ***Head align target***. If you press this button, the center of the green sphere will be
6 moved toward the point after the target.

When you are finished with assigning kinematics, you could select ***set start*** and ***set finish*** to indicate at which step the start is and at which steps it will be finished. Then you can select ***save path info***. The kinematic UOT file will be created and all information regarding the laser locations, orientation and discretization will be saved on that file.

UOT Kinematic Auxiliary tools (1/3): path on CAD

- With ***delete the current path*** you remove the current path
- With ***make new path*** you accept the current setting for the definition of a new path
- ***keep wound path*** will reserve representation of the previously designed path
- ***Save tape mesh*** will restore the INP file format mesh the check box near this button we'll show the mesh representation
- ***Size of the guide plane*** is the size of the auxiliary plane for defining the path. This plane emits rays toward the cat model to find the intersections and create the trajectory of the path. You can change the size of the guide plane manually via the edit box.
- The guide plane will be controlled with six ***XYZ movement and rotation sliders***. You could change the orientation and the locations of the guide plane.
- You can change the number of the guide planes with the guide lines.
- You can change the ***search planes*** to improve the efficiency in case of a large complex CAD file. If you are using GPU you don't need to concern about the search phases because it is so efficient.
- ***Factor DX trajectory*** defines extra discretization along the guide lines. It will divide the points between two consecutive guide lines There are several modes for the definition of the guide line. It could be either plane or carved geometry to define the flat path or curved path.
- You can also use ***pattern-making*** with this icon. The pattern-making function is still under development.
- You can save the setting of these edit boxes and sliders with ***Save sliders***. Then you will use it again easily for the next analysis.
- You can also load sliders with ***Load sliders***. Then the settings and the values of the sliders the guide plane size and the orientations of the guide points will be directly implemented.

Using these auxiliary tools, you could define every kinematics path on every CAD model with different curvatures and geometrical conditions.

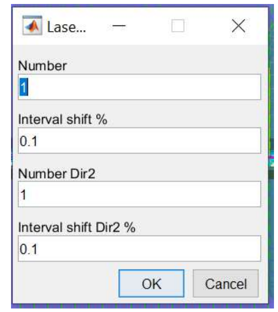


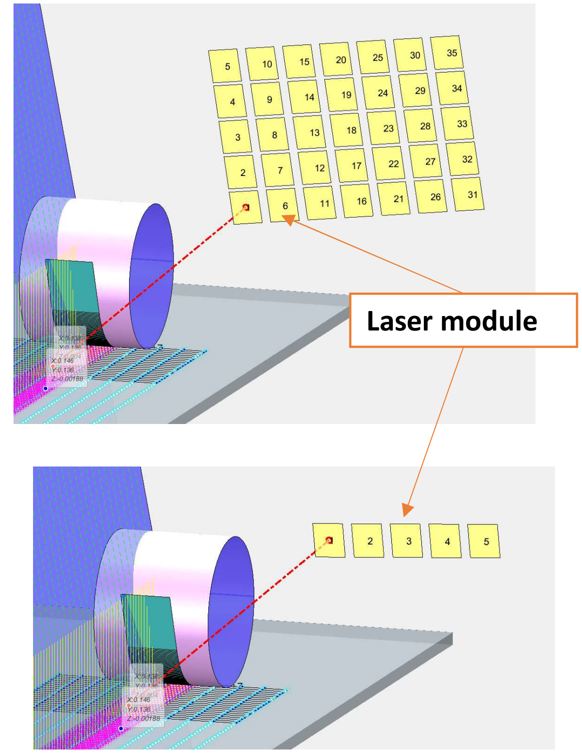
UOT Kinematic Auxiliary tools (1/3): Add laser modules

With the ***Add laser modules*** in the UOT kinematic, you could distribute different laser module rectangular shapes along Dir1 and Dir2 directions. Each of these rectangular laser heads can have its own laser power. The number after each rectangular shape will be shown in the center of the rectangles. The other possibilities that you can do with these rectangular modules are mentioned below:

- You can enter the number of laser modules in the first and second directions via the numbers.
- With interval shifts in directions 1 and direction 2, you could set the distance between two consecutive laser rectangular shapes.
- You could rotate all of this laser modulus via laser head rotation.
- The laser power and the laser intensity could also be changed in the post-processing optical and postprocessing thermal analysis. For example, if you have 4 modules of the laser, you could sit there for the numbers of the total laser power pair module.

Using these auxiliary tools, you could define different laser configurations on every model with different curvatures and geometrical conditions. This could be also used for modeling the VCSEL laser properties.

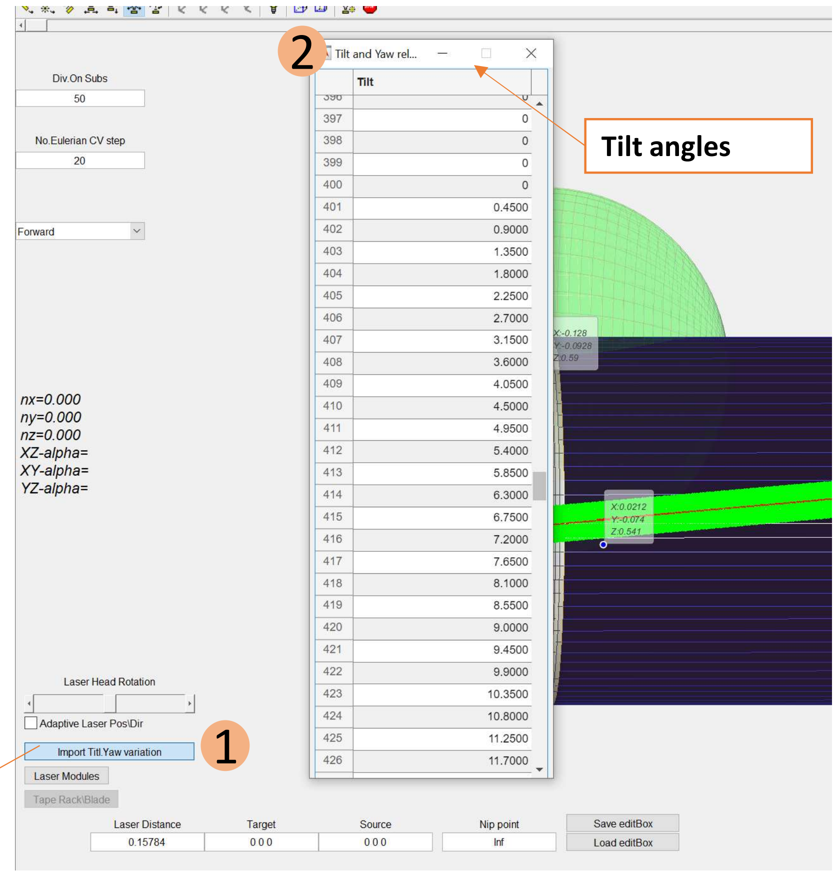




UOT Kinematic Auxiliary tools (2/3): Tilt and Yaw relative variations- Tape Rack\Blade

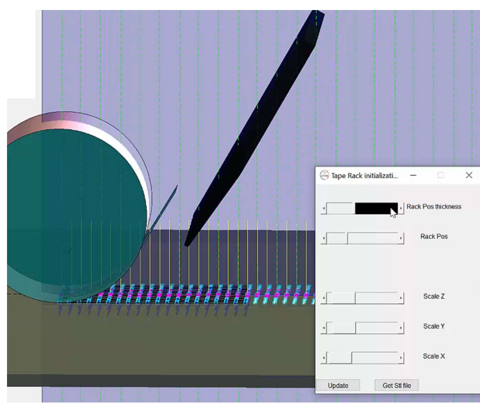
With the ***import Tilt and variation*** button in the UOT kinematic, you could import the variations of the tilt and Yaw angle from the laser head configuration at each step.

- When you click on the import tilt your variation button you could select either a TXT file or an Excel file. If your data has one column it will be only tilt angle and if your data has two columns the first column would be tilt and the second column would be Yaw relative angle variation.
- You could then select the import to bring all data into the OTOM environment.
- You could then later change the values after tilt and Yaw angles.
- When you are finished with editing the data you could close the window. The data automatically adjusts to each movement and step of the laser head configuration.
- To understand the direction of the tilt yeah, you could simply click on laser head rotation. The letters indicating the tilt and Yaw direction will appear on the laser rectangular shape.
- Please note that the number of angle steps should match the number of the steps of the roller.
- The unit of the tilt and Yaw variations are in degree.



With the ***Tape Rack\Blade***, you could import extra objects in front of the tape. This object usually helps the setup and the tape feeding system for extra adjustment but at the same time it has some influences on the optical absorption of the rays and reflections. You could import the STL file from the get STL file button. Then you could adjust its positions and the dimensions along the XY&Z axis.

The changing of the positions will be done along the directions of the incoming tape along its movement direction and along each thickness direction. To move this, you could use the slider rack position thickness and rack position sliders in the tape rack initializations. Please note that these functions and object is only available for optical analysis using GPU



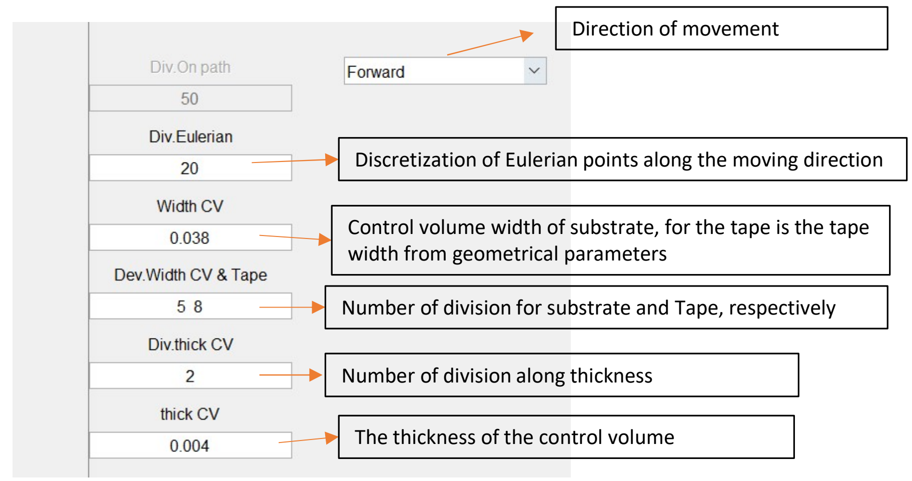
UOT Kinematic Auxiliary tools (3/3): control volume setting

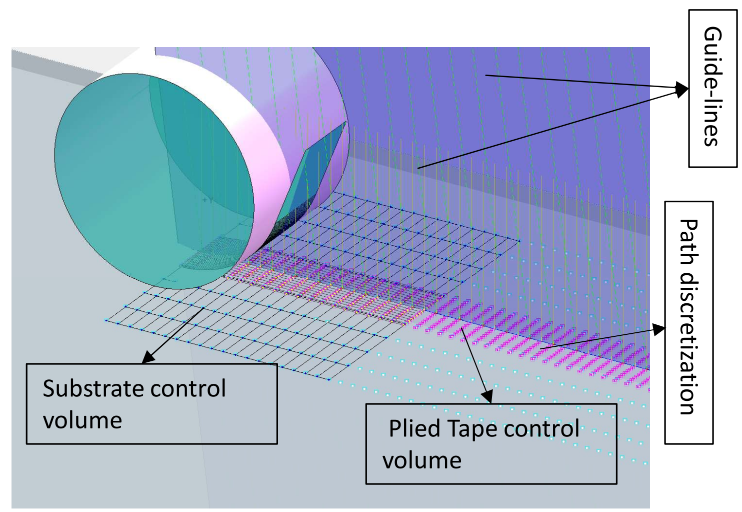
With the control volume setting you could define the discretization of the path and the UOT kinematic.

- You could select the number of Eulerian divisions, which is basically useful for the local global simulation using the UOT thermal, you could assign the size of the control volume of the substrate with the width CV, and you could select the divisions of the substrate control volume and the ply tapes, also division after thickness control volume, and finally the thickness of the control volume of the substrate.
- You can also set the direction of movement for example if the ruler starts from left to right or from right to left this could be done via forward arrow and backward arrow.

You could see the ply tape control volume and the substrate control volume on the path in the below picture. The pastor school discretization referred to the steps between the two consecutive rows of nodes along the movement of the roller. As it is seen guidelines decide about the movement of the roller on the path. These notes are equally distributed along the path.

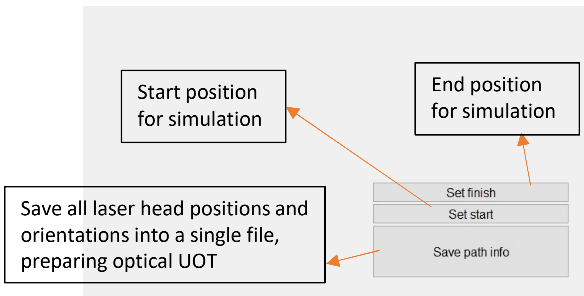
** Some of these values are not in use for the COT analysis as the COT analysis uses the global thermal modeling approach. Only the local irradiation model and consolidations using the definitions of the control volumes in the local irradiation area under the substrate.

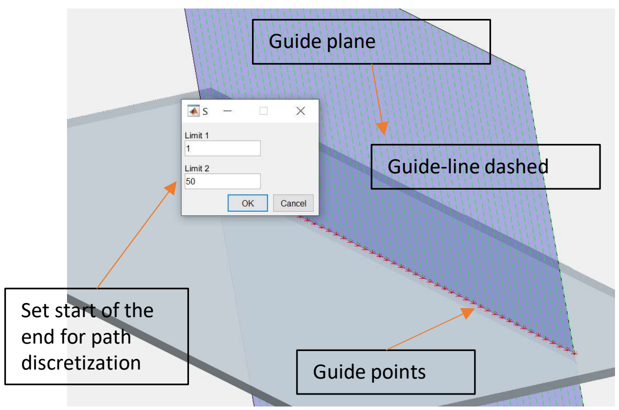


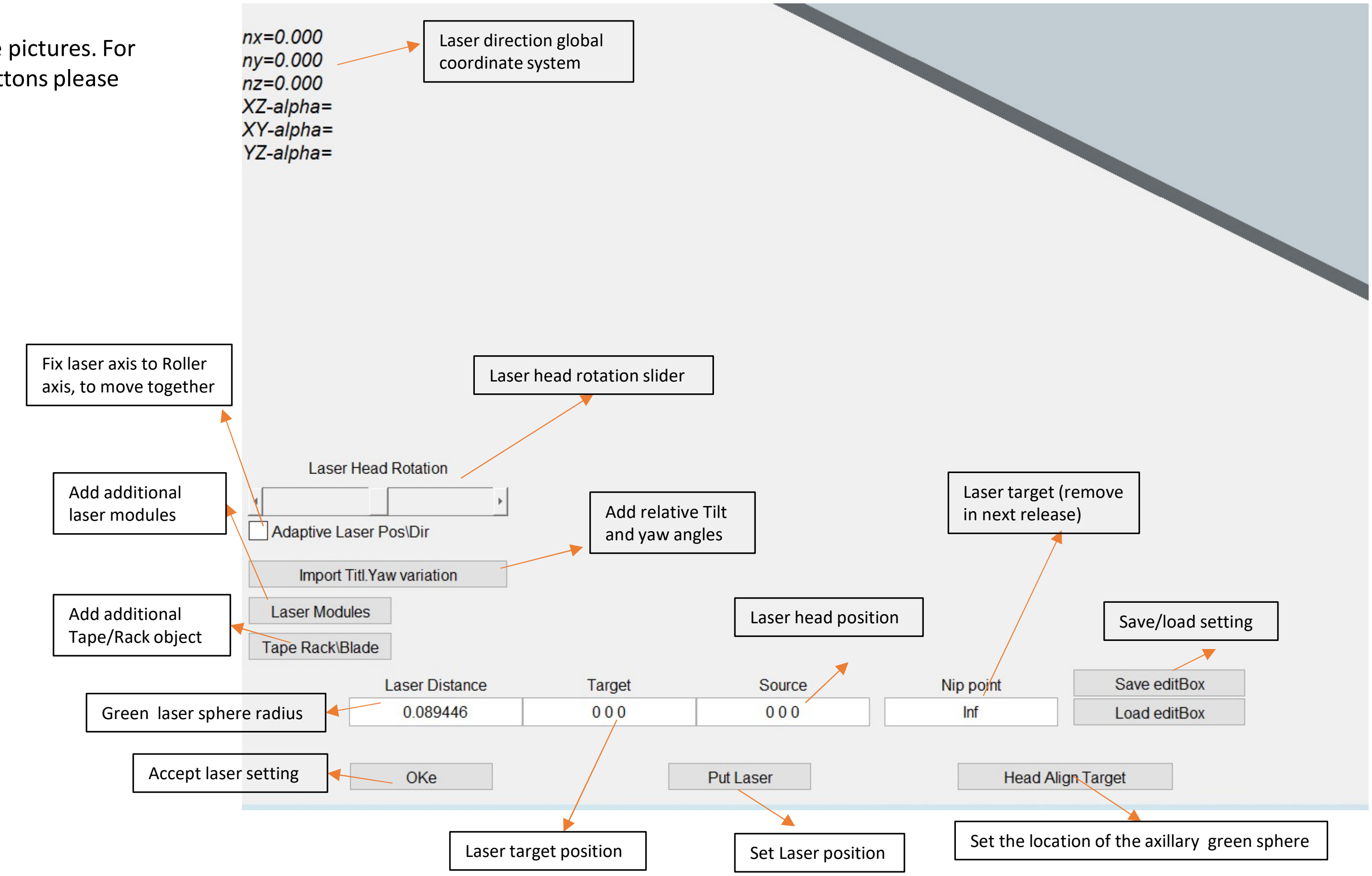


UOT Kinematic: further annotations

Other annotations in the UOT kinematics are shown in these pictures. For further assessment or explanations or functions of these buttons please refer to the tutorial videos.

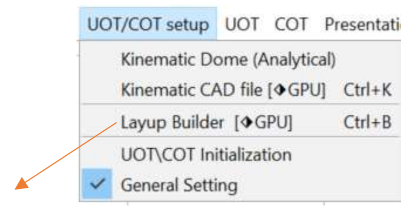






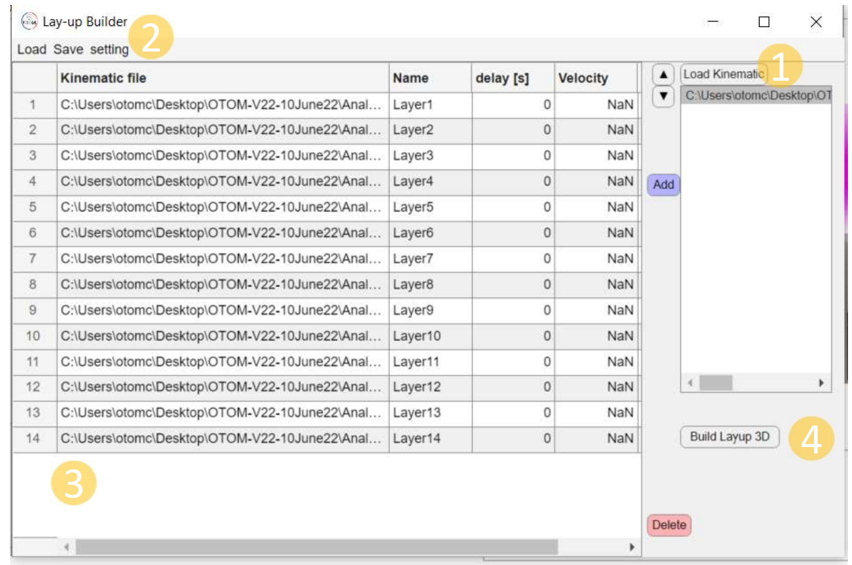
Lay-up Builder Window

With the layout builder, you design and determine the layup based on the kinematics. You could first import the kinematics that you just made with UOT kinematics. Then you could add them to the table. Based on the order that you made, it creates different layers. You could assign the layer number name delay between the layer's velocity of each layer and the total power after each layer. You could then create the 3D layout build. And check the performance after layout with the mechanical model built in OTOM software.



1 Click on load kinematics in the layup builder window to import the kinematics. The kinematics file will be added today in I screen at the right side of the layout builder. You could click on add or double-click on the names at the right side of the window to add them to the table which is on the left side of the layout builder window. You could use the arrow up or bottom to move the layers and change the orders. You could also use the delete button to delete the select it layer from the table.

2 From the menu of the layout builder you could load and save the layup. Click on the load button and then select the STL file of the base tooling and INTP mesh file after base tooling to import them in the layout build. You need these two files to build the 3D layups. If you already have the layup from the previous work you could click on load and then load layout. Then they lay up with the layers and the settings will be loaded on the layout builder window and there would be a thick box near the load layup in the menu. You could then save the layup after modifying the layout table. Finally, you could change the setting from the layout builder window. This setting is for the estimations of the boundary of the layers for creating the boundary conditions and 3D geometry of the layup. It usually you don't need to change those values, only if you have a curved and complex shape you might need to increase the curvature value which is between zero to 1.



The table shows the layup order after the kinematic file, you could double-click on the name and change the name of the layer, and you could also change the delay between the layers. If you set the velocity or power as nan, the velocity and power from the processing parameter will be used.

4 When you finish setting up the layers in the table and you upload their STL base after tooling and the mesh file after tooling you could create the 3D representation of the layout. click on the build layup 3D button, to represent the layup creation. In the 3D layout window, you could then see all the information and geometry overlaps of the layers. You could hide and show the layers and change the lighting of the scene. You could also perform the contact mechanical analysis which shows the structural performance based on the force applied to the structure. Please refer to the appropriate section for the mechanical analysis of the layup.
Please note that if you have a higher number of layers and a higher number of nodes the creation of the layup 3D might take longer. The reason is due to the fact that all of the boundary conditions and the node-to-node interactions among the layers will be created.

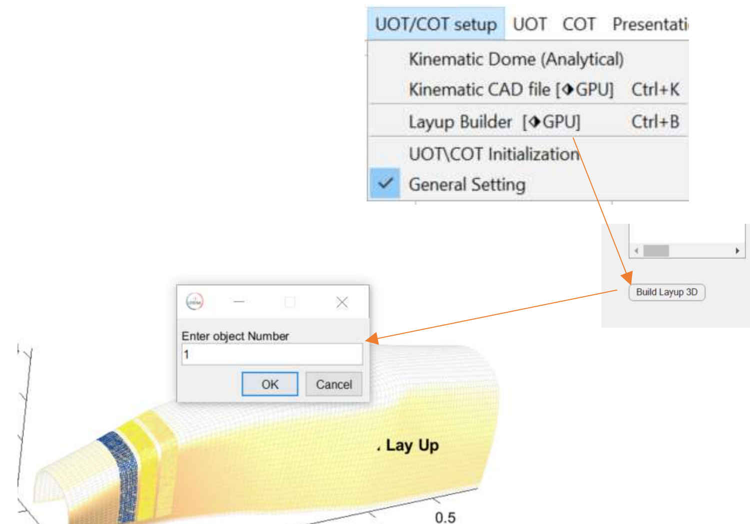
Lay-up Builder: 3D layup

When you represent a 3D layup with Layup Builder, a window will show up showing all the layers. There are show\hidden and lightning buttons on the right side of the figure. There is also a built-in mechanical contact model inside this window.

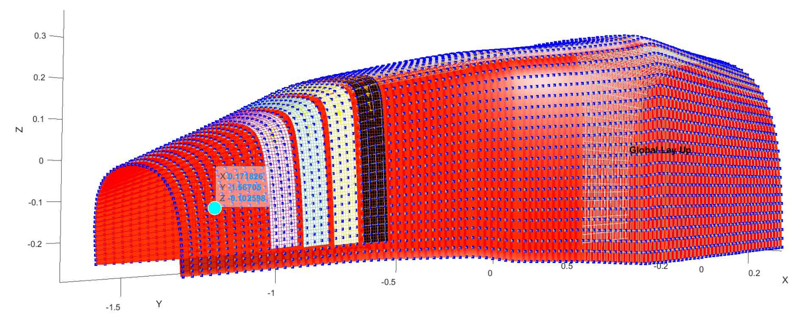
To perform mechanical analysis, click on the boundary condition icon or select its function from ***FEM mechanical analysis.***

1. Insert the object number and click on OK to set the boundary conditions on that object.
2. You have some options to set the displacement boundary conditions. The first one is single point boundary conditions where you can fix X, Y, and Z axis systems for one node. Or you can select the set of nodes based on the tangent relation between the nodes. Click on the ***Tangent angle*** and select a point on the surface. All other points which have a tangent value with the continuous limitations of angle (*default 18 deg*) will be selected. You will see the highlighted nodes during the preview. You could then add them as new boundary conditions and set the corresponding displacement. You could set a displacement as a reference fixed point or as a loading boundary condition. (Please check tutorial videos for more details).
3. For force boundary conditions, you can also click on the external force icon or selected it from ***FEM mechanical analysis*** menu. You could then apply external force on a specific node or on a set of nodes. For selecting a set of nodes, you could select tangent angles or closest points. As explained earlier, with the tangent angle all of the nodes which has a continuous tangent limitation tolerance will be selected. For the close points, the first closest point which has a tangent tolerance will be selected. For details on the selections of these boundary conditions and performing the FEM mechanical analysis, please refer to the tutorial videos.
4. Finally, when you set the boundary condition and the material model you could select the ***Solver*** from ***FEM mechanical analysis*** menu. The FEM solver with run based on the number of time steps that you inserted in the software. When the solving is finished, it shows the displacement and deformation quantities of the 3D layup. You could then observe the results by selecting the ***Results observer*** from the menu. You could represent the 3D contours of deformations along the 3-axis, strain values, and stress values like Von misses equivalent stress.

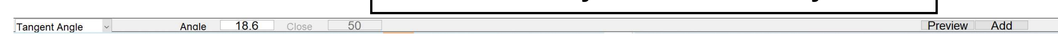
The contact FEM model for a composite layup is based on the penalty method. The node-to-node contact was enriched in the assembled stiffness matrix to impose the perfect contact. It assumes that the layers are perfectly connected to each other and transferred energy and forces. For further technical support please refer to the technical notes of the software.



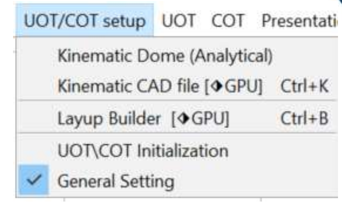
Object Selection for BC or External forces



Node Selection for BC or External forces



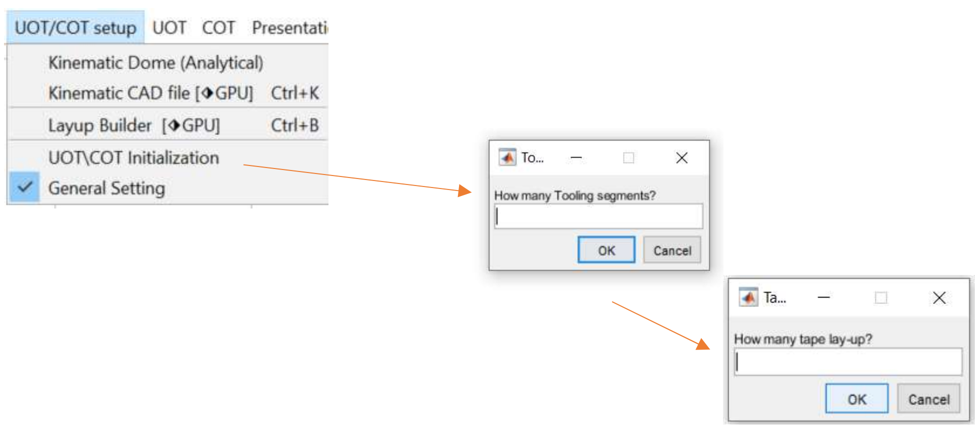
UOT/ COT initialization



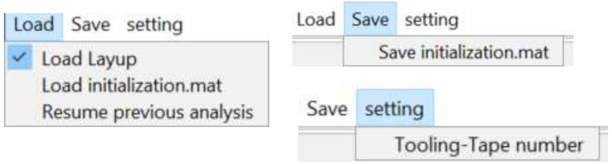
With UOT/COT initialization, the assembly of the layers could be built up.

- **Assembly**: The user can define how many tooling layers are available, for example, the presence of the thin Kapton layer or some steel tooling or any layers at the beginning of the process. Based on the order layup the geometry will be assembled.
- **BC**: The boundary conditions among the layers will be defined, for example, how much is the heat resistance between each layer and what is the condition of the roller consolidation. If a layer has a fixed temperature or initial temperature different than others, it could be assigned.
- **Material**: the material will be defined, if a part of the mesh has a different material or has a void (material does not exist), all could be defined.
- **Lay-up pattern (only COT)**: if different kinematics is used for the analysis of the COT model, the Layup should be loaded first into the window. For that purpose, the Layup builder in the software should be defined first. In the case of COT analysis, the delay between layers will be fetched from the lay-up build, The number of the Tape will be also acquired from the Layup builder.

Click on **UOT/ COT setup > UOT/ COT initialization** button to set the number of Tooling and Tapes and start UOT/ COT initialization.



UOT/ COT initialization

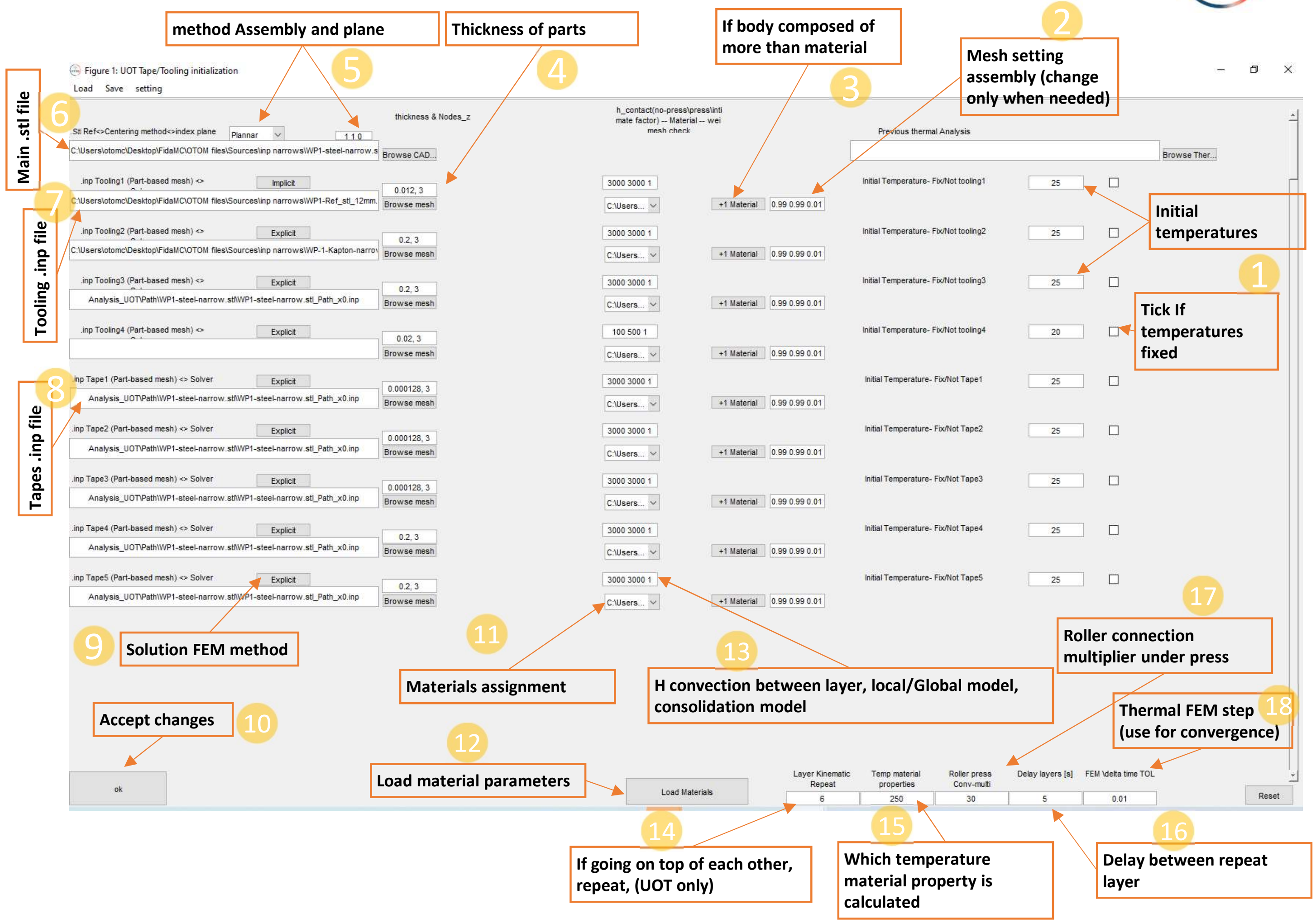


With the menu of UOT initialization, you could ***load, save*** and change the setting regarding the initializations after UOT and COT models. From the ***load tab***, you could load the layout. When you do this the number of the two links and tape will be created based on the lay-up file. If you wish to change the tooling number for example adding some insulation layer or captain layer you could use the setting and ***change the number of tape and tooling*** and add extra tooling objects. You could also ***load the previous initialization UOT or COT*** files. Then the setting of the previous initialization file will be loaded. If you like to resume the analysis from the previous conditions, you could select the ***resume previous analysis***. The function is not still in use.

When you update all of the settings of this window you could click on save to keep the current setting and save it as a file.

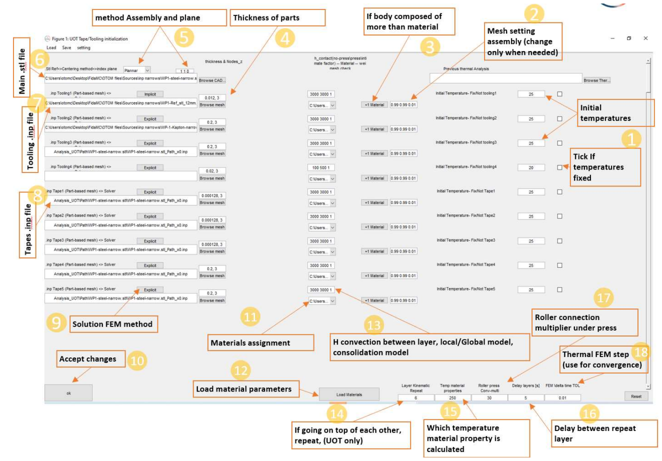
From the setting, you have options to change the number of Toolings and tapes. When you update this the number of the field boxes on the main window of the initialization will change.

The rest of the buttons of this window are numbered and the function related to those numbers are explained below.



Confidential

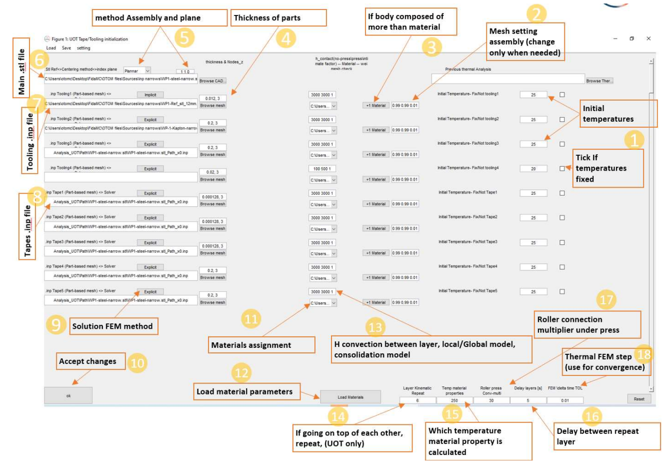
UOT/ COT initialization



1. On the right side of the figure you see boxes for initial temperature value and also a tick box to indicate that this temperature could be fixed during analysis.
2. You can change the setting of the mesh assembly here these values are used specifically for carved surfaces they helped the software to decide and create the shallow area around the notes to create boundary conditions between the objects. The first two values are weight factors, and it is recommended not to be changed. The last value which is between zero to 1 it is used for curvature definition. The higher number tells the software that they should consider a higher margin on the carved locations. This value multiplies the thickness value and checks if there would be any object on that region for considering boundary conditions and layout build.
3. If your layer or tooling consists of more than one material or if some part of the tape or tooling has void it means there is no material you can click on plus one material. Then base it on there material set that you created during meshing you could indicate that that part of the mesh has different materials. Please note that the mesh file should have a set name with the name Mat-1, -2,…
4. The thickness and number of divisions for the thickness can be interred here. UOT thermal model uses this value for local and consolidation models this will not be in use for COT models.
5. Method assembly is used to indicate where the other Toolings will be placed on the base tooling. If you have two links based on the cartesian axis, like a flat panel, you could write the plane that these two objects will be centered. For example, plane 110 indicates that they will be centered at XY plane. If you have a curved geometry and axial configuration like a tube or vessel you could use the axial option. Then the two links will be centered based on the axial configuration for example center of the two tubes will you place at the same location.
6. This is the main STL file you can use borrows CAD to select the main STL file.
7. This is the tooling mesh file which has INP format. You can use borrows mesh to select the mesh file.
8. This is the Tape mesh file which has INP format. You can use borrows mesh to select the mesh file.
9. This is the solver of the finite element method. You can select between explicit and implicit methods. For a larger number of nodes, the stiffness matrices become larger and the inverse matrix became more time-consuming in the implicit method. For a large number of nodes, it is recommended to use explicit methods. But in general, implicit methods give more accurate results than explicit methods. For details of the implicit and explicit please check the references.
10. The OK button accepts all of the settings of the initialization UOT and COT. If you click on this button, it will also close the window.
11. With the pop-up menu which is located at the Center for each object you can select the material for each object. Please note that this material should first load it.
12. With the load material button you could select the material that you created before in the software. When you do load materials the content of pop up menu at the middle will be updated.
13. This is the convection coefficient after local global models. The first value is the convection coefficient in local irradiation model and global FM model. The second one is the convection coefficient in the consolidation model. The third value is the multiplier for the consolidation model. This value will be removed in the future release.
14. For UOT analysis, you can indicate the layout to be repeated via this textbox.

UOT/ COT initialization

15. With this text box you indicate at what temperature the material parameters are considered it for unsteady optical thermal simulation. here the default value is 250 degrees Celsius.
16. The roller press convection coefficient multiplier is used to multiply the consolidation coefficient. It will be used in the local consolidation model and also in global COT models. It is basically multiplied by the value that was indicated in process parameters as the tape roller convection coefficient.
17. The delay layer indicates how much delay would be between two consecutive layers. It will be used for your OT analysis. In COT analysis the delay layers will be from the layout table.
18. The FM delta time tolerance indicates the time step for the global finite element model. If your convection coefficient is high you need to select lower values as delta time here. Indicating higher delta time FEM, you might get a divergence in the result which are not correct. Thus it is always better to keep it as low as possible. Choosing a lower value for F and delta time will increase the time as steps and hence the computational cost will increase.



Finally, when you select the reset button at the right corner of the window all of the edit boxes will be erased.

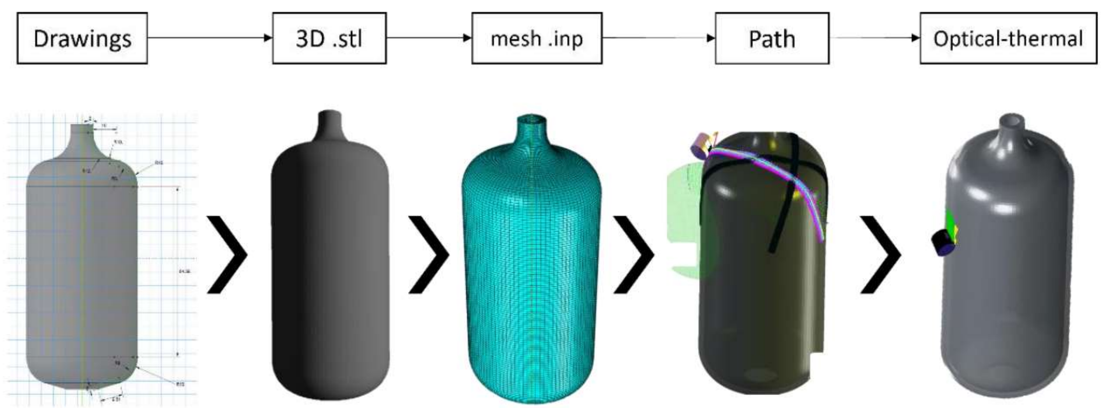
When you finish initializing the values and accept the changes, after asking you for the analysis name it will be used as the name for UT analysis and the directory will be created and the path of the optical and kinematic analysis. This name will not be in use for COT analysis.

Furthermore, when you accept the changes, the software checks if the INP mesh files need extra,

UOT/ COT procedure summary

UOT stands for Unsteady Optical Thermal Simulation and Continuous optical thermal Simulation. For performing UOT/COT analysis, the steps should be followed as below:

1. Make the geometry which has a thickness, 3D solid body (based on the .stl binary)
2. Create .inp format mesh of the base geometry using software like Gmesh, Abaqus or Salome using quad 8 nodes mesh, (Assign up and bottom surface and save it as set). You can also import nodes and mesh connectivity into OTOM and edit them and make sets via Utility functions.
3. Import the CAD stl file in OTOM and create the kinematics and path
4. Create UOT/ COT initialization including all layers conditions (Create Lay-up builder before this step for COT analysis)
5. Perform the UOT or COT analysis (For UOT, perform optical, pre-thermal simulation before the thermal analysis).

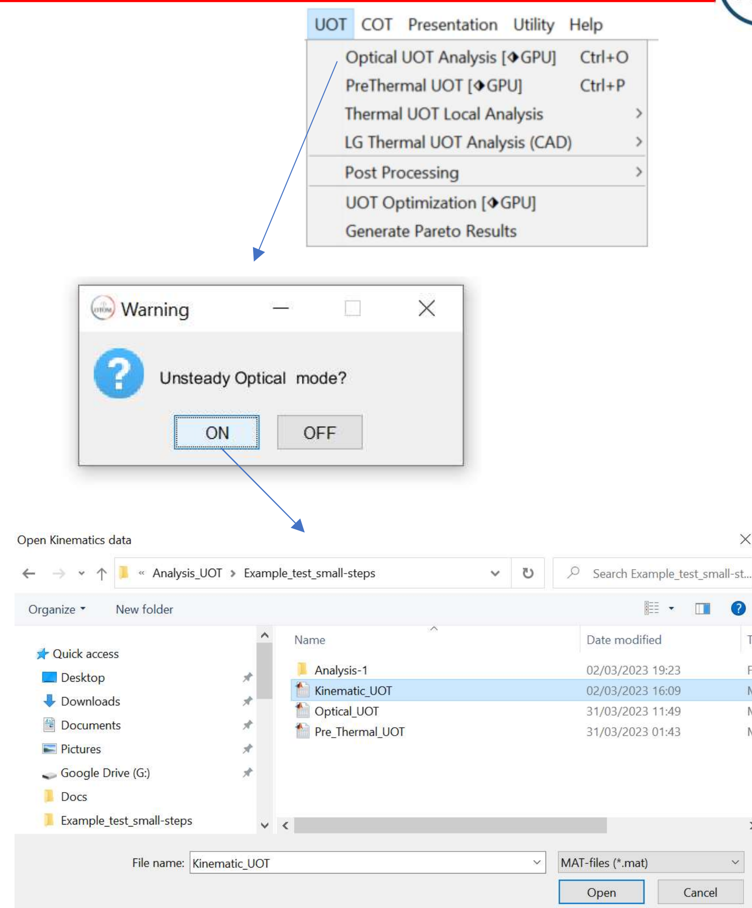


Optical UOT

From ***UOT> Optical UOT analysis*** you can select optical UOT analysis.
With this function, you could perform optical analysis based on the unsteady optical thermal modeling for the analytical geometry of tube vessels and domes as well as the CAD models.

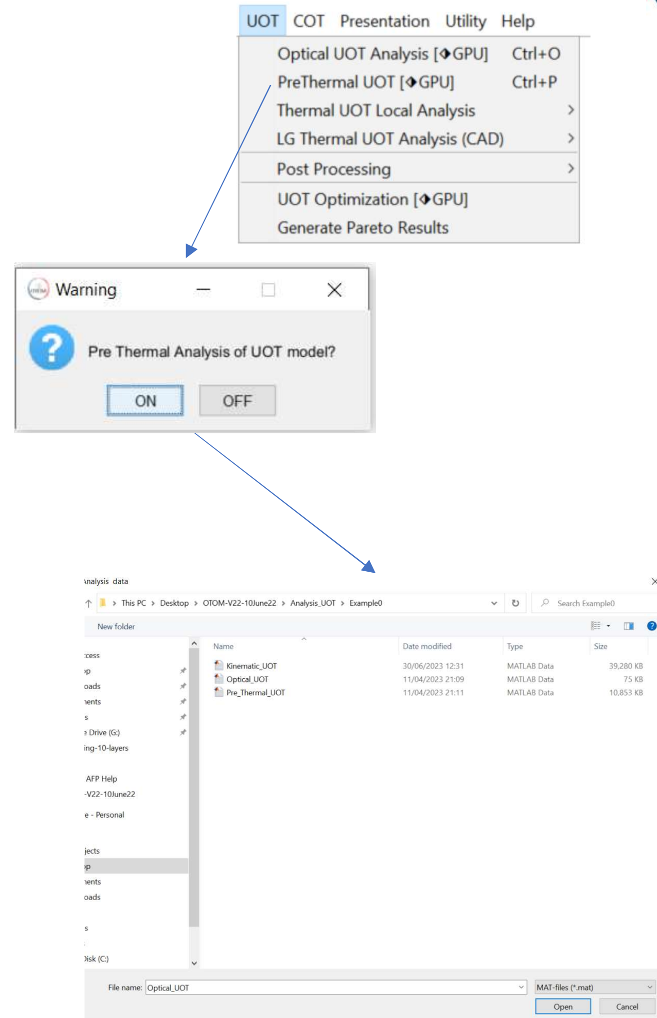
- This function works based on the CPU and the GPU. First, you need to select the UOT kinematic file that you created before. Then you could select other settings like laser divergence. At this moment you cannot change the number of the rays or discretization of the laser head. All this information was saved during the UOT kinematic creation.
- With these functions, you set up the optical analysis to be performed at each kinematic step. When you select the kinematic file, the calculate button will change to calculate UOT optical to indicate that the optical analysis is ready to be run. This mode does not support BRDF anisotropic reflection function.
- After performing this analysis optical UOT file will be created in the corresponding job name directory.
- During running the analysis, text files related to the intersection of the rays with the object will be created.
- To turn off the optical UOT analysis, you could reselect this item optical UOT analysis and select the off button. Then the calculate function button in the main software window will change to the normal mode.
- You can further analyze the data via post-processing optical analyzers or fear OTOM AFP viewer.

***Please check the below reference for further details of the optical models:*
https://www.mdpi.com/1996-1944/13/11/2449
https://www.sciencedirect.com/science/article/pii/S0264127520306651



Prethermal UOT

From ***UOT> Prethermal UOT*** you can select Prethermal UOT analysis. This function distributes the optical node to the thermal nodes on the substrate control volume. This means that the intensity distributions on the tape or substrate could be calculated based on this function. This function only is in use for UOT thermal analysis and it's not for COT thermal analysis. In COT all of this procedure of the optical and prethermal model will be done in the COT loop.



- This function works based on the CPU and the GPU. First, you need to select the UOT optical file that you created before.
- With these functions, you set up the optical nodes to be distributed at each kinematic step. When you select this function, the calculate button will change to calculate UOT Prethermal to indicate that the Prethermal analysis is ready to be run.
- After performing this analysis Prethermal UOT file will be created in the corresponding job name directory.
- To turn off the Prethermal UOT analysis, you could reselect this item Prethermal UOT analysis and select the off button. Then the calculate function button in the main software window will change to the normal mode.

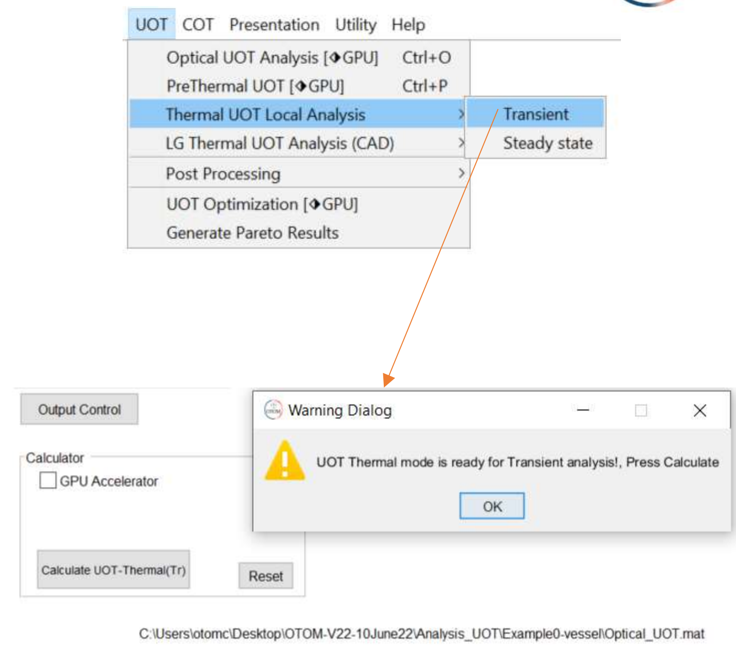
***Please check the below reference for further details of the Prethermal models:*
https://www.mdpi.com/1996-1944/13/11/2449

Confidential

Thermal UOT: Analytical

From ***UOT> Analytical Thermal UOT*** you can select thermal UT analysis for analytical on the vessels and domes. This model does not have a global temperature estimation. This means that all of the temperature predictions will be calculated only for the local irradiation area. Thus there would not be any overlaps of the layers and complex layup configurations for this analysis mode. This model works based on the CPU. Some other information for this model is listed below:

- If your geometry curvature is almost the same during the roller movement, you can select a steady-state thermal analysis. It basically solved the problem in only one step. This is the case when you have a constant curvature geometry, and the conditions of the temperature remain the same.
- If you have a variable geometry for example from the transition of the vessel to domes you need to perform transient thermal UOT analysis.
- The total time of the simulation's time steps is determined based on the distance of the consecutive step (from UOT kinematic) and also the velocity that you provided via the processing parameters.
- When you select this analysis of UOT thermal, the calculate button on the main window will change to calculate UOT thermal. This means that the software is ready to start the simulations by pressing the calculate button.



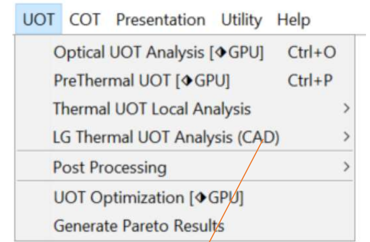
*** Please check the below reference for further details of the UOT thermal models:*
https://www.sciencedirect.com/science/article/pii/S0264127520306651

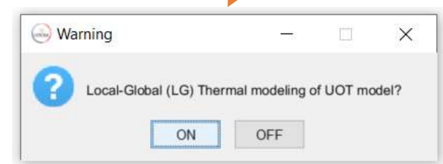
Thermal UOT: CAD-based

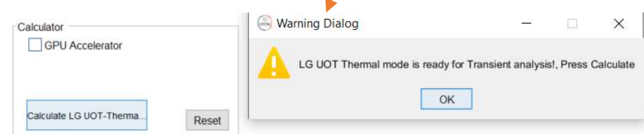
From ***UOT> CAD Thermal UOT*** you can select thermal UT analysis for any CAD geometry. This model has a global temperature estimation to calculate not only the local irradiation area but also the consolidation phase as well as the cooling phase on the whole geometry. However, overlaps of the layers are limited to repeating the layers on top of each other (check COT for complex layup). This model works based on the CPU and GPU. Some other information for this model is listed below:

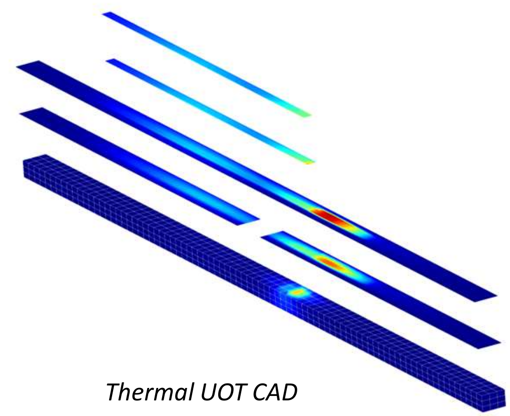
- This analysis works based on the combination of the local irradiation thermal finite difference model, consolidations finite difference model, and global finite element thermal model. The transferring of the data among these models will be calculated and operated in each step.
- You could get the information on the irradiation area, temperature along the nip points temperature along the movement kinematics, and temperature along the thickness at the nip point as the standard outputs
- you can post-process the thermal data in the complementary software of OTOM AFP viewer.
- The total time of the simulation's time steps is determined based on the distance of the consecutive step (from UOT kinematic) and also the velocity that you provided via the processing parameters.
- When you select this analysis of UOT thermal, the calculate button on the main window will change to calculate UOT thermal. This means that the software is ready to start the simulations by pressing the calculate button.
- For complex layup configurations, please refer to COT thermal analysis.

***Please check the below reference for further details of the UOT thermal models:*
https://www.sciencedirect.com/science/article/pii/S0264127520306651







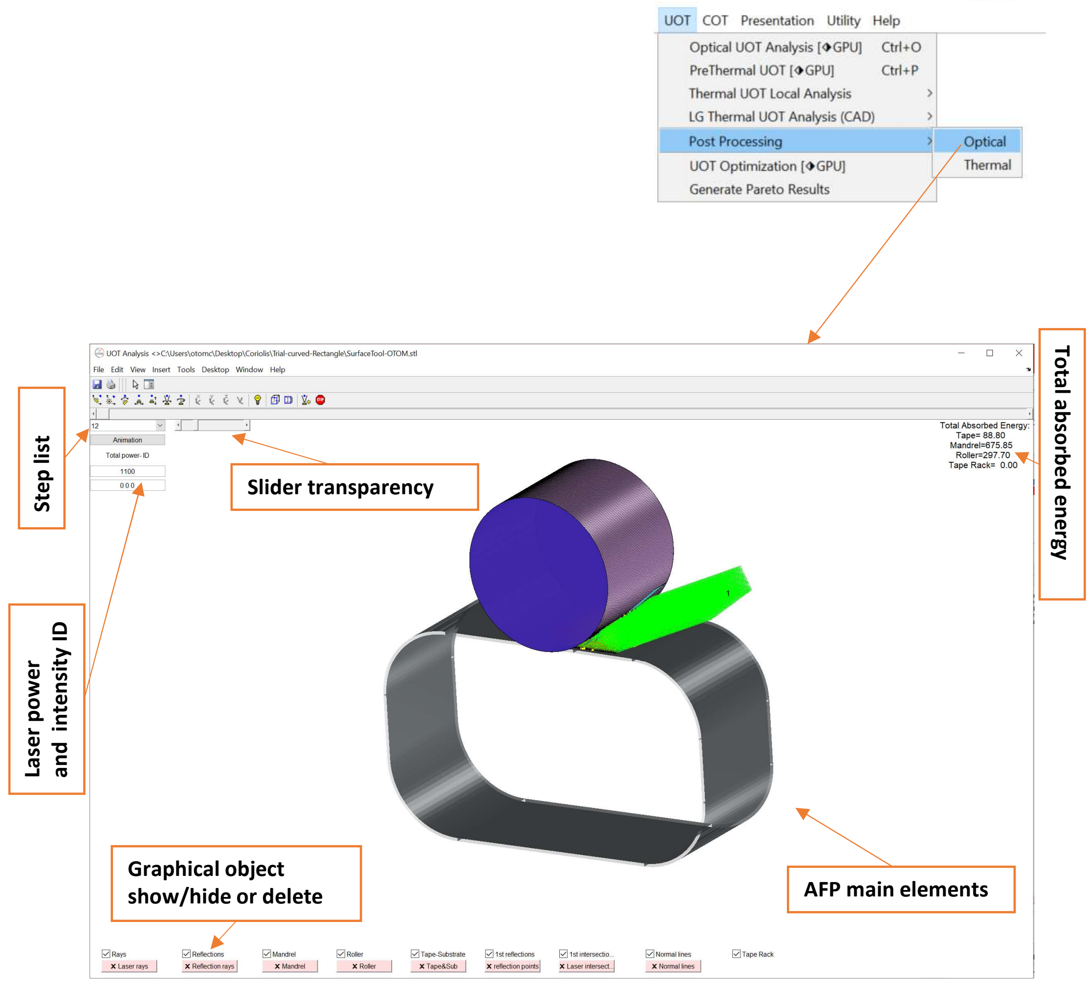


Thermal UOT CAD

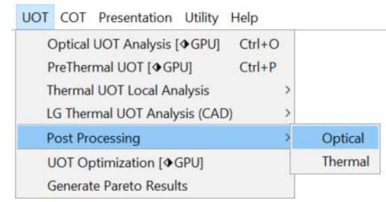
Optical post-processing UOT

With optical postprocessing, you can observe more details of the optical analysis. You can see all of the intersection points all of the rays all of the reflections and exactly the whole geometry in one single shot. More details of this window are listed below:

- it has slider transparency to change the transparency of the rays and reflections
- total absorbed energy text to show the total absorber power per object for tape mandrel roller and tape rack if existed
- Established a step slider to move the different steps. You can press and select the steps or move the steps by a slider and the roller moves along the path at the same time the raise and reflection will be updated.
- With animation bottom you could generate animations in MP4 format and it will be saved in the directory of the analyzer's name.
- You can insert different laser power. When you insert a new laser power you should move the steps and then the whole setup and absorber power values will be updated.
- In case you have a several laser modulus for example if you have a four laser module you could insert four different values for total laser power.
- Below the laser power text box there is a text box for laser ID. you could insert the laser intensity value there. You can find more information about the laser ID intensity in the laser distribution control. You can choose different intensities like uniform intensity top hat intensity Gaussian intensity and the complex 4D Gaussian intensity distribution.
- You can hide and show the different objects with the tick box and add the below side of the window. You can also delete those objects by pressing the cross icon of each object. This will be useful when you have a lot of data points for example lots of rays, and your system has some struggle to represent and see the movement through the steps. Just to make it more affordable for your system unfeasible to have a smooth experience of representation of the optical analysis, you could delete those objects from the scene.
- Finally you could also use the mouse and the keyboard shortcut to rotate and zoom in or zoom out on the AFP main element. You can find more information regarding using the mouse or keyboard shortcut intersection related to the mouse and keyboard interaction.

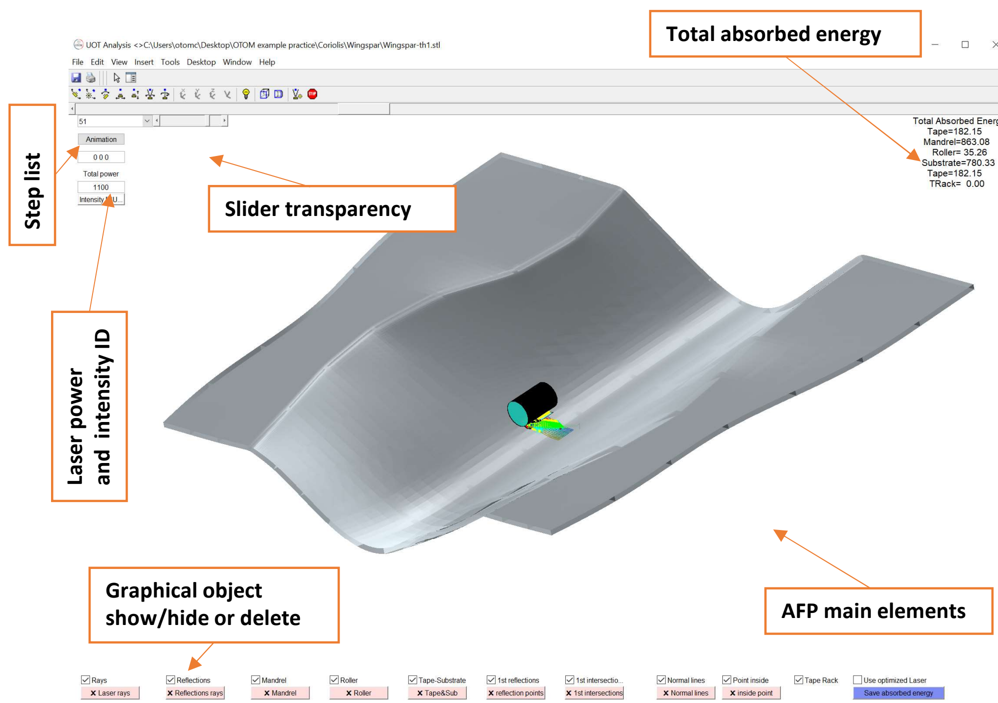


Thermal post-processing UOT

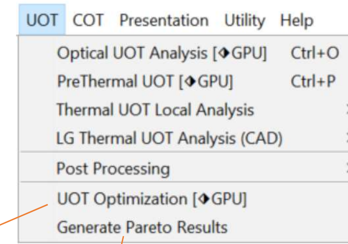


With thermal postprocessing, you can observe more details of the optical analysis and Pre-thermal models, and also analytical UOT thermal. You can see all of the intersection points all of the rays all of the reflections and exactly the whole geometry with temperature (if applicable) in one single shot. More details of this window are listed below:

- it has slider transparency to change the transparency of the rays and reflections
- total absorbed energy text to show the total absorber power per object for tape mandrel roller and tape rack if existed
- Established a step slider to move the different steps. You can press and select the steps or move the steps by a slider and the roller moves along the path at the same time the raise and reflection will be updated.
- With animation bottom you could generate animations in MP4 format and it will be saved in the directory of the analyzer's name.
- You can insert different laser power. When you insert a new laser power you should move the steps and then the whole setup and absorber power values will be updated.
- In case you have a several laser modulus for example if you have a four laser module you could insert four different values for total laser power.
- Below the laser power text box there is a text box for laser ID. you could insert the laser intensity value there. You can find more information about the laser ID intensity in the laser distribution control. You can choose different intensities like uniform intensity top hat intensity Gaussian intensity and the complex 4D Gaussian intensity distribution.
- With the intensity distribution you could represent the absorbed intensity power on the tape and substrate Please note that you need a NVIDIA GPU for this function.
- You can hide and show the different objects with the tick box and add the below side of the window. You can also delete those objects by pressing the cross icon of each object. This will be useful when you have a lot of data points for example lots of rays, and your system has some struggle to represent and see the movement through the steps. Just to make it more affordable for your system unfeasible to have a smooth experience of representation of the optical analysis, you could delete those objects from the scene.
- Finally you could also use the mouse and the keyboard shortcut to rotate and zoom in or zoom out on the AFP main element. You can find more information regarding using the mouse or keyboard shortcut intersection related to the mouse and keyboard interaction.



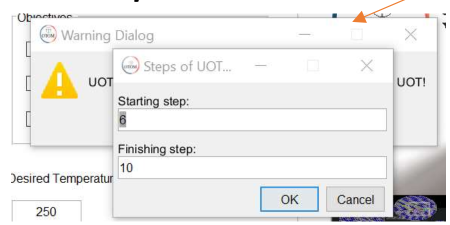
UOT Thermal post-processing: Optimization



From ***UOT*** tab and then ***UOT optimization*** you can access the function for optimization of the out model. This function basically worked for UOT analytical model. If you click on this function it shows the starting and the finishing steps for the optimization process. You could enter that information and then paste it into the optimization window on the main software window, you can perform optimizations based on the unsteady optical thermal modeling.

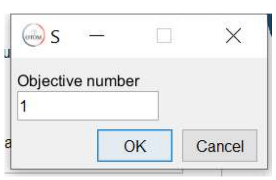
***Please check the below reference for more information regarding this function:*
https://www.sciencedirect.com/science/article/pii/S0264127520306651

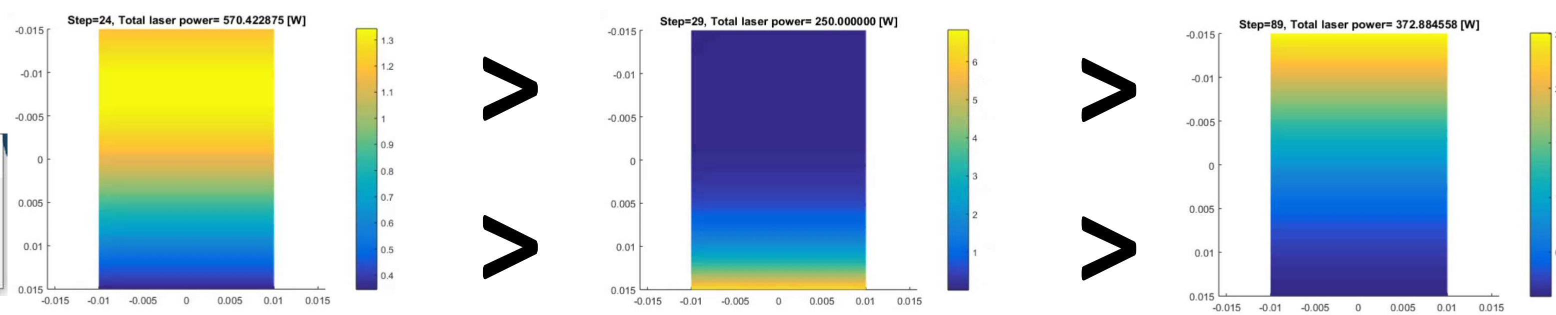
UOT optimization



When the UOT optimization is concluded and when you have a multi-objective optimization, you could use the function to generate pareto results. It shows the result based on the selected objective and also generates animations after optimizer data

Generate Pareto results

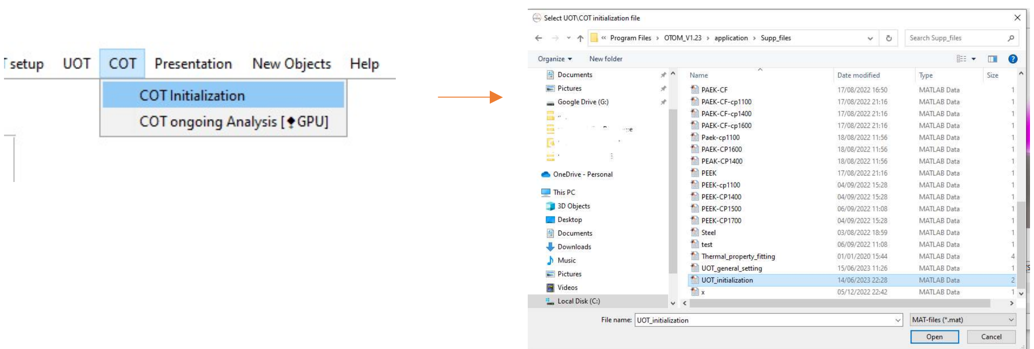


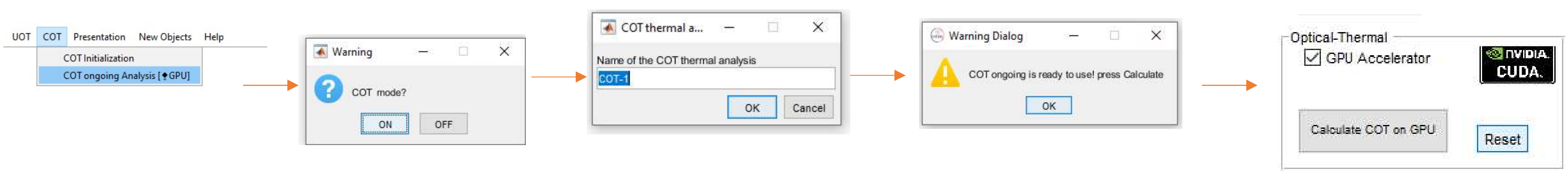


COT thermal **(Only With CUDA)**

In COT, both Optical and Pre-thermal will be executed together with thermal analysis, all at the same time. Having assigned UOT/COT initialization and Layup build, the COT analysis is ready to go.

1. Click on the COT initialization window and select the UOT/COT initialization. Mat.
2. Click on the COT ongoing analysis to set the name and prepare it to run
3. The Calculate button name will change to COT mode.
4. You can now run COT.

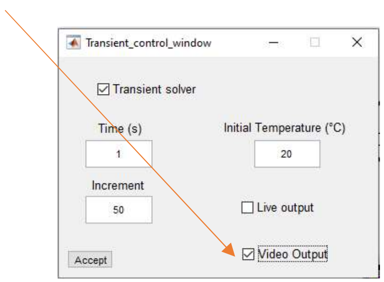


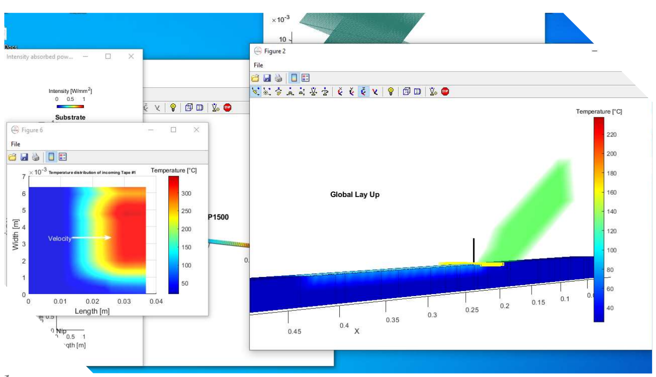


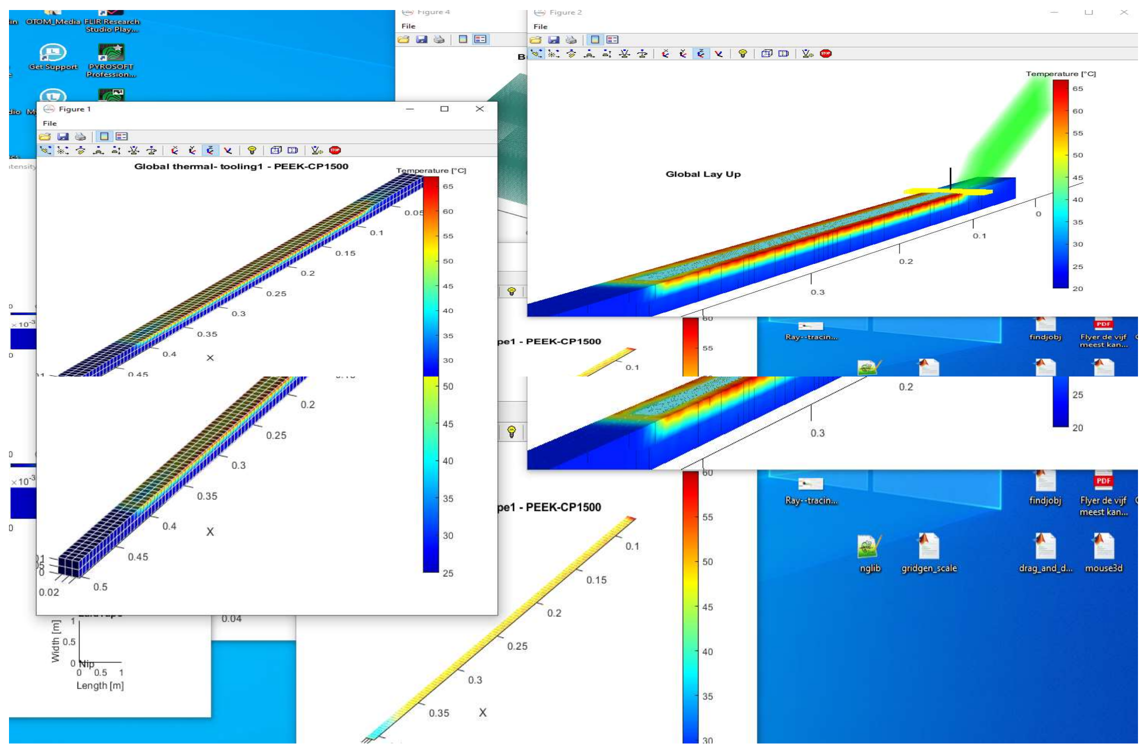
Run and Results

When the COT simulation is running, several figures will show up during the simulation progress including Global layup progress, thermal temperature window of each object, the temperature at the nip and substrate, and the heat flux absorbed power intensity. For COT some plot may be different than UOT during the simulations.

You can record the animation during the analysis. To record the animation, go to the ***Transient analysis window*** and Tick the “Video output” box.



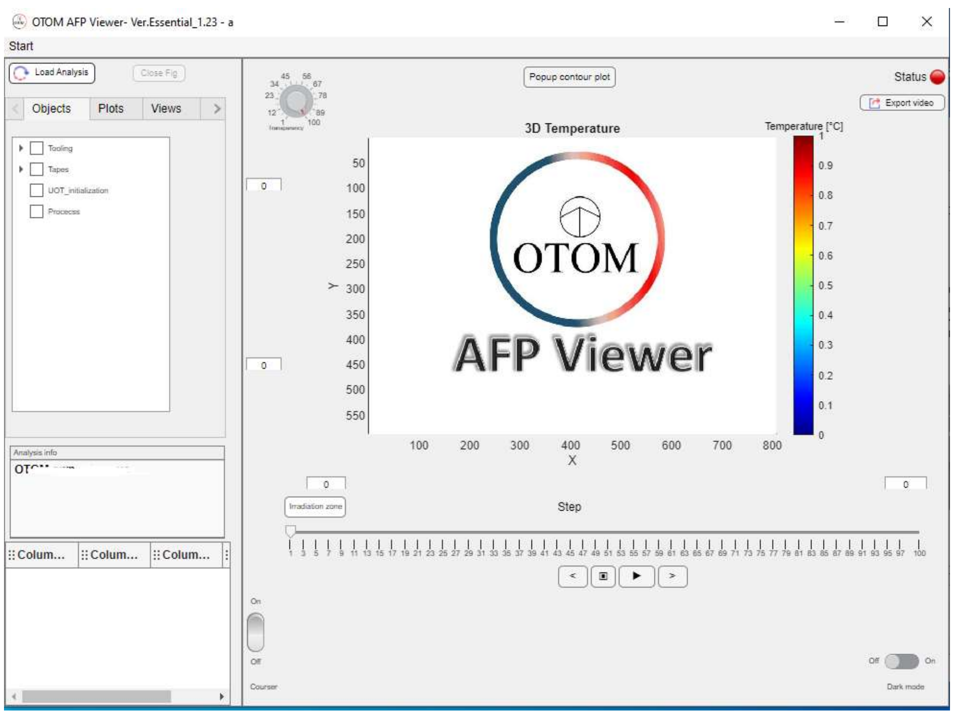


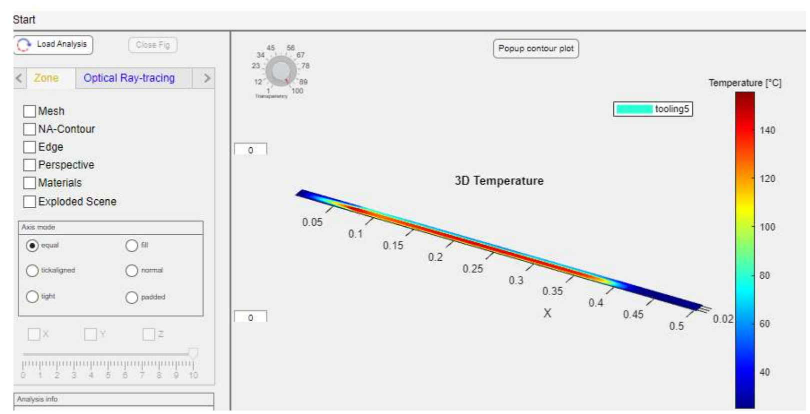


Post-process thermal analysis

With OTOM AFP Viewer software, you can load both UOT and COT analysis results for further investigation.

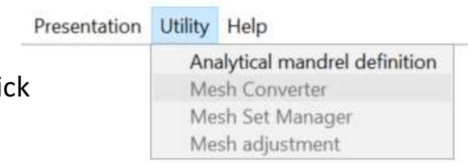
Use OTOM AFP Viewer to extract data from the layers, generate animations, exploded views, relative Crystallinity prediction, and many more.





**See OTOM Viewer document for more details

OTOM Utility mesh tools

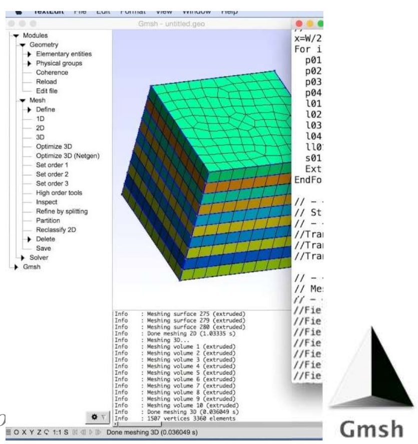


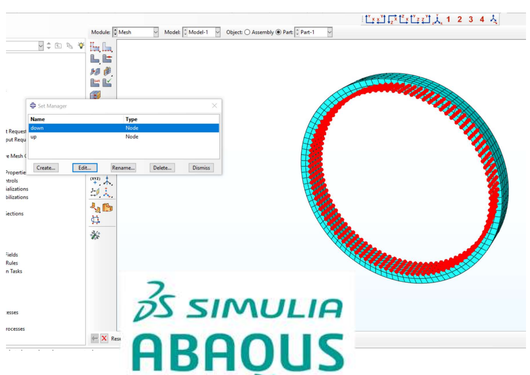
With OTOM utility mesh tools, you can adjust and convert your mesh data. Currently, the OTOM AFP software only supports Brick 8-node 3D mesh.

Based CAD model: The base mesh can be generated based on the.Stl file. OTOM support.Inp mesh which can be generated using software like Abaqus, Gmsh, Salome, etc.
Plied Tape mesh: For creating a mesh of the plied tape, you can use external software, or you can use UOT kinematics in the OTOM AFP software which creates the mesh based on the plied nodes.

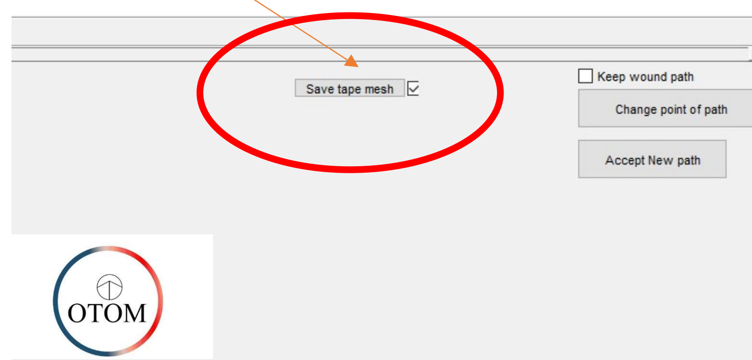
The functionality of mesh adjustment in OTOM is still under development.

**It is very important that all upper layer nodes will be collected and labeled as “up” or “Top”, so then the OTOM software knows where the new tapes will be plied, The same procedure should be done for “bottom”, “down” surface,
**For the tape the similar procedure just the geometry should be thinner, but at a similar position, and orientation!
** You can also create a set of elements, named Mat-1, and Mat-2,… Within the same mesh, then assign second, third,… Material or Void





Creating mesh in OTOM



END

For suggestions and feedback please contact:
info@otomcomposite.eu