The European Materials Modelling Council

# In Power: Design and simulation of advanced materials for innovative mirrors suitable for use in different type of CSP plant

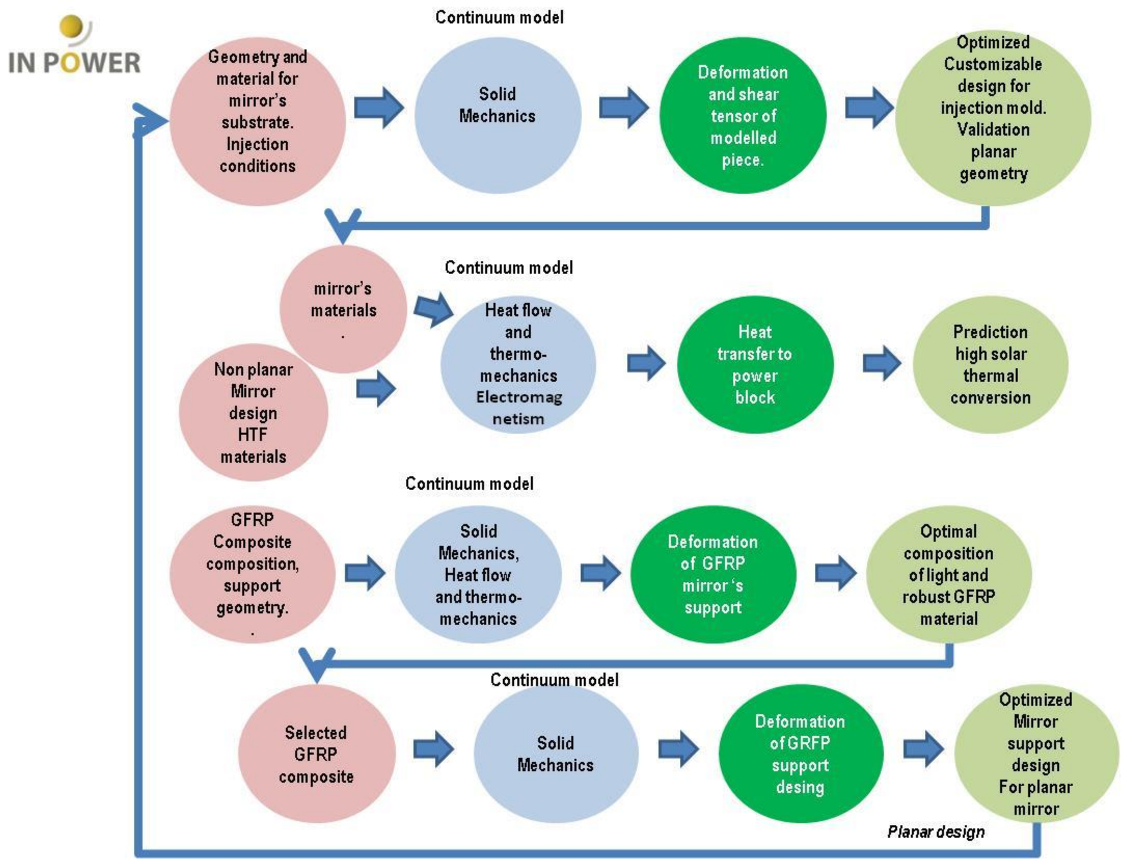
*Version date: 2016.10.19*

**Purpose of this document:**
Definition of a data organization that is applicable to ALL materials modelling simulations. The fiche should contain all elements that are needed to describe a simulation. This information spans from the end-user (manufacturer) information to the computational modelling details.

<table>
<tr><th colspan="4">THE SIMULATION GENERAL DESCRIPTION</th></tr>
<tr><td>1</td><td>USER CASE</td><td colspan="2">Design and simulation of advanced materials for innovative mirrors suitable for use in different type of CSP plant. The materials include polymeric substrate, metallic reflective coating, self-healing coating, antidust coating and composite for support. This includes also the manufacturing process and the operational conditions.</td></tr>
<tr><td rowspan="4">2</td><td rowspan="4">CHAIN OF MODELS #1</td><td>MODEL 1</td><td>Continuum model for industrial application. Solid mechanics. (model 4.1 in ROMM) The model 1 is dedicated to find the optimal processes in order to avoid deformation in the final customizable CSP mirror substrate; the validation will be done using planar geometry.</td></tr>
<tr><td>MODEL 2</td><td>Continuum models:. Tightly coupled Heat flow and thermo-mechanics. (model 4.3 in ROMM) and Electromagnetism (optics) (model 4.6 in ROMM)<br>Define the optimal non planar mirror geometry in order to find a solution in which there is a four times increase of the power generation considering performance of materials in service</td></tr>
<tr><td>MODEL 3</td><td>Continuum model for industrial application. Solid mechanics. (model 4.1 in ROMM)<br>Continuum model. Heat flow and thermo-mechanics. (model 4.3 in ROMM)<br>Definition of new composite for mirror support</td></tr>
<tr><td>MODEL 4</td><td>Continuum model for industrial application. Solid mechanics. (model 4.1 in ROMM)<br>Definition of support geometry considering performance of materials in service.</td></tr>
<tr><td>3</td><td>PUBLICATION</td><td colspan="2">Not available</td></tr>
<tr><td>4</td><td>ACCESS CONDITIONS</td><td colspan="2"><u>Restrictive licence</u>: Autodesk Moldflow, Altair HyperWorks, Altair Partner Alliance softwares, TracePro and Matlab, ABAQUS from SIMULIA Platform</td></tr>
</table>

Each model used in a simulation is documented in four chapters:
1-Aspect of the User Case/ system simulated with the model
2-Model
3-Computation
4-Post-Processing. In some cases between two simulations the post/preprocessing takes place. This processes the output (or part of the output) of one simulation into input for next simulation.

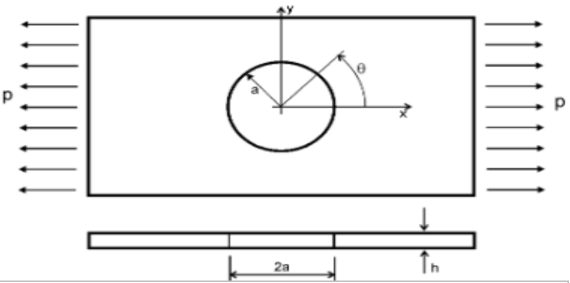
**Workflow for models needed for In Power, Chain 1.**

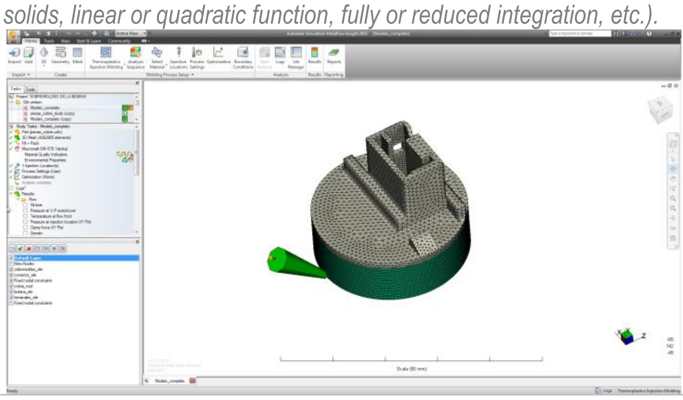


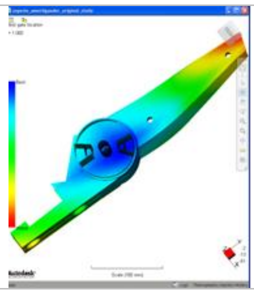

1. **MODEL 1 for Design of injection mold for mirror substrate for CSP. [WP3, Task 3.1 and 3.2]**

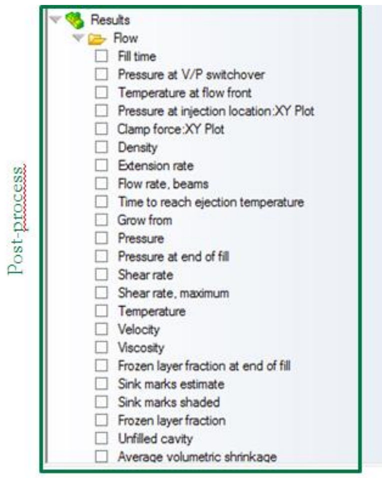
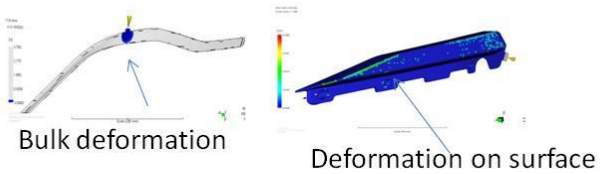
| 1 | ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED | |
|---|---|---|
| 1.1 | ASPECT OF THE USER CASE TO BE SIMULATED | ***The model 1 is dedicated to find the optimal injection mold process in order to avoid deformation on customizable CSP mirror substrate.*** |
| 1.2 | MATERIAL | *Polycarbonate based material.* |
| 1.3 | GEOMETRY | *Planar geometry, from 10cm x 10cm to 0.7m x 1m, multilayer material system. Customizable to other geometries. Design of mirror's substrate by CATIA from task 3.1.* |
| 1.4 | TIME LAPSE | *Static setting.* |
| 1.5 | MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS | *Manufacturing process: temperature and pressure of injection process* |
| 1.6 | PUBLICATION | |

| 2 | GENERIC PHYSICS OF THE MODEL EQUATION | | |
|---|---|---|---|
| 2.0 | MODEL TYPE AND NAME | *Continuum Solid mechanics. (model 4.1 in ROMM)* | |
| 2.1 | MODEL ENTITY | *The entity in this materials model is finite volumes corresponding to polymeric substrate injected with different process parameters.* | |
| 2.2 | MODEL | **Equations** | ***Hooke law relating forces and displacement*** |

| | | | |
|---|---|---|---|
| | **PHYSICS/ CHEMISTRY EQUATION PE'S** | **Physical quantities for each equation** | ***Lagrangian deformation and strain tensor. Shear strain***<br>*Position of material point, deformation due to external forces* |
| | **MATERIALS RELATIONS** | **Equations** | $\sigma_r = \frac{p}{2}\left(1-\frac{a^2}{r^2}\right)\left[1+\left(1-3\frac{a^2}{r^2}\right)\cos 2\theta\right]$<br>$\sigma_\theta = \frac{p}{2}\left[1+\frac{a^2}{r^2}-\left(1+3\frac{a^4}{r^4}\right)\cos 2\theta\right]$<br>$\tau_{r\theta} = -\frac{p}{2}\left(1-\frac{a^2}{r^2}\right)\left(1+3\frac{a^2}{r^2}\right)\sin 2\theta$<br> |
| | | **Physical quantities/ descriptors for each MR** | *Variables: shear ($\tau$) and strain ($\sigma$) due to external loads during injection process.*<br>*Parameters: (1) external forces applied during the injection process (ex. P in previous scheme) (2) dimension of polymeric sample (ex. a, h in previous scheme) (3) material density, etc.* |
| **2.4** | **SIMULATED INPUT** | | |
| **2.5** | **PUBLICATION** | *Publication documenting the model.* | |

| | | | |
|---|---|---|---|
| **3** | **SPECIFIC COMPUTATIONAL MODELLING METADATA** | | |
| **3.1** | **NUMERICAL SOLVER** | *FEM ( Finite Element Method) and CFD (Computacional Fluid Dynamic)* | |
| **3.2** | **SOFTWARE TOOL** | *Autodesk Moldflow* | |
| **3.3** | **TIME STEP** | *If applicable* | |
| **3.4** | **COMPUTATIONAL REPRESENTATION** *Refers to how your computational solver represents the material, properties, equation variables,* | **PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL** | *At this stage, a first approximation of the original geometry is done by a set of points (nodes) connected via elements. Modelling of the deformation of geometry, meshing, and element definition (shells or solids, linear or quadratic function, fully or reduced integration, etc.).*<br> |
| | | **BOUNDARY CONDITIONS** | *Visual representation into the geometry. Ex. pressure injection profile.* |

| | | | |
|---|---|---|---|
| | | |  |
| | | **ADDITIONAL SOLVER PARAMETERS** | *They will be defined during the project, when facing the simulation analysis of the mirror' substrate mold.* |
| **3.5** | **PUBLICATION** | *Publication documenting the simulation or web link* | |

| **4** | **POST PROCESSING** | |
|---|---|---|
| **4.1** | **THE PROCESSED OUTPUT IS CALCULATED FOR** | *The optimal parameters, such as temperature at flow front, pressure at injection location, flow rate, for injection process and mold geometry in order to avoid deformation in bulk and in surface of mirror's substrate. The outputs of the modelling give the following values regarding injection process. Optimal design and process will define the final geometry of injection mold for manufacture the samples. In particular, planar samples will be empirically validated.*<br>*This will be used in model 2.*   |
| **4.2** | **METHODOLOGIES** | *Uneven compaction piece volumetric contractions leading to very large deformation are determined by visual representation of internal deformation of the pieces The deformation can be visualized in bulk and in surface. Then, if deformations exist in simulated piece, change in parameter of injection process (such as type of injection process, injection points, etc,) will be done in order to avoid them.*   |
| **4.3** | **MARGIN OF ERROR** | *To be defined.* |

**2. MODEL 2 for simulated new solar collector with non planar mirrors for CSP. [WP6, task 6.4, linked with Task 6.3 and 6.5].**

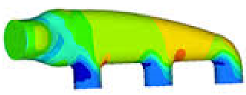
| **1** | **ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED** | |
|---|---|---|
| **1.1** | **ASPECT OF THE USER CASE TO BE SIMULATED** | *Define the optimal non planar mirror geometry in order to find a solution in which there is a four times increase of the power generation considering performance of materials in service* |
| **1.2** | **MATERIAL** | *Mirror's materials* |

| | | |
|---|---|---|
| | | Heat Transfer Fluids properties.<br>Heater composed material |
| 1.3 | GEOMETRY | Non planar geometry for mirror<br>Heater container geometry |
| 1.4 | TIME LAPSE | Static state |
| 1.5 | MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS | In service conditions: incident solar radiation<br>Different incident angle and incident power radiation will be used. |
| 1.6 | PUBLICATION | |

| 2 | GENERIC PHYSICS OF THE MODEL EQUATION | | |
|---|---|---|---|
| 2.0 | MODEL TYPE AND NAME | Tightly couples system of<br>Continuum model. Heat flow and thermo-mechanics. (model 4.3 in ROMM)<br>Continuum model. Electromagnetism (optics) (model 4.6 in ROMM) | |
| 2.1 | MODEL ENTITY | The entity in this materials model is finite volumes. | |
| 2.2 | MODEL PHYSICS/ CHEMISTRY EQUATION PE'S | Equations | Head transfer balance equation of:<br>Advection: $Q = v\rho c_p \Delta T$<br>Conduction: $Q = v\rho c_p$<br>Convection: $\dot{Q} = hA(T_a - T_b)$<br>Radiation: $Q = \epsilon\sigma(T_a^4 - T_b^4)$<br><br>Incompressible Navier-Stokes equations<br>$$\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} - \nu\nabla^2\boldsymbol{u} = -\nabla w + \boldsymbol{g}$$<br>Snell's equations<br>$$\frac{\sin\theta_1}{\sin\theta_2} = \frac{v_1}{v_2} = \frac{\lambda_1}{\lambda_2} = \frac{n_2}{n_1}$$ |
| | | Physical quantities for each equation | For CFD models:<br>$R_i$ is the equation residual at an element vertex $i$,<br>$W_i$ is the weight factor,<br>$Q$ is the heat flux conservation equation expressed on an element basis and<br>$V^e$ is the volume of the element.<br>$h$ is the heat transfer coefficient,<br>$A$ is the area of the object,<br>$\sigma$ is the Stefan-Boltzmann constant<br><br>For Ray-tracing models:<br>$L_o$ is the outgoing light,<br>$L_e$ is the emitted light and<br>$\omega$ is the reflected light.<br>$\theta_i$ angle measured from the normal of the boundary,<br>$v_i$ velocity of light in the respective medium,<br>$\lambda_i$ wavelength of light in the respective medium |

| | | |
|---|---|---|
| MATERIALS RELATIONS | MR Equations | Thermal expansion<br>$$\alpha_V = \frac{1}{V}\left(\frac{\partial V}{\partial T}\right)_p$$ |
| | Physical quantities/ descriptors for each MR | Where<br>$V$ is the volume,<br>$T$ is the temperature,<br>subscript $p$ indicates that the pressure is held constant,<br>$\rho$ is density |

| | | |
|---|---|---|
| | | $c_p$ heat capacity at constant pressure<br>$\epsilon$ is the emissivity<br>$n_i$ refractive index of the respective medium<br>. |
| 2.4 | **SIMULATED INPUT** | *Optimized mirror materials and the final quality of the surface's piece, from model 1, in order to be use as interface between mirror and air in Ray Trace model.* |
| 2.5 | **PUBLICATION** | *Publication documenting the model.* |

| 3 | **SPECIFIC COMPUTATIONAL MODELLING METADATA** | | |
|---|---|---|---|
| 3.1 | **NUMERICAL SOLVER** | *MonteCarlo, Ray Trace Model, C Finite Element Method. Each model is independent and it is used as an input for the next model.* | |
| 3.2 | **SOFTWARE TOOL** | *Altair HyperWorks, Altair Partner Alliance softwares, TracePro and Matlab* | |
| 3.3 | **TIME STEP** | *. It will be determined during the creation of the model, depend on itsconvergency.* | |
| 3.4 | **COMPUTATIONAL REPRESENTATION** *Refers to how your computational solver represents the material, properties, equation variables,* | **PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL** | *Finite element-based Computational Fluid Dynamics (CFD)*<br>$$R_i = \iiint W_i \, Q \, dV^e$$<br>*Where*<br>$R_i$ *is the equation residual at an element vertex* $i$,<br>$W_i$ *is the weight factor,*<br>$Q$ *is the heat flux conservation equation expressed on an element basis and*<br>$V^e$ *is the volume of the element.* |
| | | **BOUNDARY CONDITIONS** | *Solar irradiation profile*<br>*as function of* $\theta_i$ , $\lambda_i$ *and* $n_i$ *refractive index of the respective medium (interface air/mirror)*<br>*Heater container profile*<br> |
| | | **ADDITIONAL SOLVER PARAMETERS** | *They will be defined during the project, when facing the simulation analysis of the heater and the heat transfer fluid.* |
| 3.5 | **PUBLICATION** | | |

| 4 | **POST PROCESSING** |
|---|---|
| 4.1 | **THE PROCESSED OUTPUT IS CALCULATED FOR** | *The out of process: Qcap : heat captured on the heater due to solar radiation and QHTF: heat transferred by the Heat Transfer Fluid is used to calculate the solar-thermal conversion (ST). We define ST as the ratio between both outputs:*<br>$$ST = \frac{Q_{HTF}}{Q_{cap}}$$ |
| 4.2 | **METHODOLOGIES** | *Solar radiation profile is used as incident spectra radiation. Using (at first step) a parabolic mirror shape, Snell equation is used in order to know the amount of radiation arrives to the heater. Then this radiation is converted into heat trough the heater's container. The heat capture by the heater* ($Q_{cap}$) *will depend on material of the heater and its geometry (both fixed). The heat then is transfer to the heat transfer fluid* ($Q_{HTF}$) *into the heater and piping system.*<br>*Knowing* $Q_{cap}$ *and* $Q_{HTF}$, *ST is calculated. It is expected to have higher ST conversion respect to PTC solar field using a fixed land size. Then the optimization of the geometry of non planar mirror will be evaluated in order to achieve high values of ST.* |

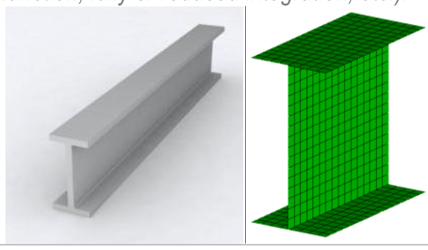
| | | |
|---|---|---|
| 4.3 | MARGIN OF ERROR | *To be defined.* |

### 3. MODEL 3 for optimization of composite for mirror structure. [WP6, task 6., linked with Task 6.2].

| 1 | ASPECT OF THE USER CASE/SYSTEM TO BE SIMULATED | |
|---|---|---|
| 1.1 | ASPECT OF THE USER CASE TO BE SIMULATED | *Definition of new composite for mirror support* |
| 1.2 | MATERIAL | *Glass Fibre Reinforced Polymer (GFRP)* |
| 1.3 | GEOMETRY | *Planar geometry of mirrors but support geometry will be defined. Standards profile will be used as starting point.* |
| 1.4 | TIME LAPSE | *Quasi-static* |
| 1.5 | MANUFACTURING PROCESS OR IN-SERVICE CONDITIONS | . <br> *In service conditions: bending forces.* |
| 1.6 | PUBLICATION | |

| 2 | GENERIC PHYSICS OF THE MODEL EQUATION | | |
|---|---|---|---|
| 2.0 | MODEL TYPE AND NAME | *Tightly coupled system of* <br><br> *Continuum model for industrial application. Solid mechanics. (model 4.1 in ROMM)* <br> *Continuum model. Heat flow and thermo-mechanics. (model 4.3 in ROMM)* | |
| 2.1 | MODEL ENTITY | *The entity in this materials model is finite volumes of GFRP with different ratio of glass fibre.* | |
| 2.2 | MODEL PHYSICS/ CHEMISTRY EQUATION PE'S | **Equations** | *Hooke Law* <br> $\{f\} = [K] \cdot \{u\}$ |
| | | **Physical quantities for each equation** | *Where* <br> $\{f\}$ *is the force vector,* <br> $[K]$ *is the stiffness matrix and* <br> $\{u\}$ *is the displacement vector.* |
| MATERIALS RELATIONS | | **MR Equations** | *Stiffness matrix for a transversely isotropic material* <br> $[K] = \begin{bmatrix} K_{11} & K_{12} & K_{12} & 0 & 0 & 0 \\ K_{21} & K_{22} & K_{23} & 0 & 0 & 0 \\ K_{31} & K_{32} & K_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & K_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & K_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & K_{66} \end{bmatrix}$ |
| | | **Physical quantities/ descriptors for each MR** | *Where* $K_{ij}$ *are different physic properties depending on the analysed case and the position into the matrix (e.g. they are computed from Young's modulus, shear modulus, Poisson's ratio and thermal conductivity among others).* |
| 2.4 | SIMULATED INPUT | | |
| 2.5 | PUBLICATION | | |

| 3 | SPECIFIC COMPUTATIONAL MODELLING METADATA | |
|---|---|---|
| 3.1 | NUMERICAL SOLVER | *Finite Element Method* |

| | | | |
|---|---|---|---|
| 3.2 | **Software tool** | *Altair HyperWorks and Altair Partner Alliance softwares.* | |
| 3.3 | **Time step** | *If applicable* | |
| 3.4 | **Computational representation** *Refers to how your computational solver represents the material, properties, equation variables,* | **Physics equation, material relations, material** | *Hooke law, defining the external loads taking into account thermo-mechanical properties of the composite such as Young modulus, inertia moment, etc. The modelling of the geometry will be done through meshing, and element definition (shells or solids, linear or quadratic function, fully or reduced integration, etc.).*  |
| | | **Boundary conditions** | *Visual representation into the geometry through HyperMesh and HyperView.* |
| | | **Additional solver parameters** | *They will be defined during the project, when facing the simulation analysis of the GFRP composite structure.* |
| 3.5 | **Publication** | *Publication documenting the simulation or web link* | |

| 4 | POST PROCESSING |
|---|---|
| 4.1 | **The processed output is calculated for** \| *The Optimal composition of GFRP for light and robust supports will be calculated. The density and then the ratio of component are selected in terms of robustness of defined pieces under external loads. The density will be selected following the in service requirements (tension, impacts, etc) depend on solar field specification of the CSP pilot plant. Among this, This output will be used for design of planar mirror support as input of next model 4* |
| 4.2 | **Methodologies** \| *Standard tests of GFRP samples will be performed in order to obtain empiric results. The same tests will be modelled using FEM. Correlation of model vs. empiric will be carried out in order to adjust simulation model. GFRP structure designs will be computed via FEM. Optimisation and validation of the GFRP structure. Variables calculated:*<br>- *Stress field of the GFRP structure below yield strength*<br>- *Strain field of the GFRP structure to accomplish service conditions*<br>- *Displacements of the GFRP structure below maximum deflections permitted*<br>- *Weight of GFRP structure optimised for minimum weight without compromising integrity of the structure.* |
| 4.3 | **Margin of error** \| *Preliminary results will be correlated with standard tests to validate the model and ensure the minimum error (expected error will be below 5-10%).* |

## 1. MODEL 4 Solid mechanics for 3D design for mirror structure.

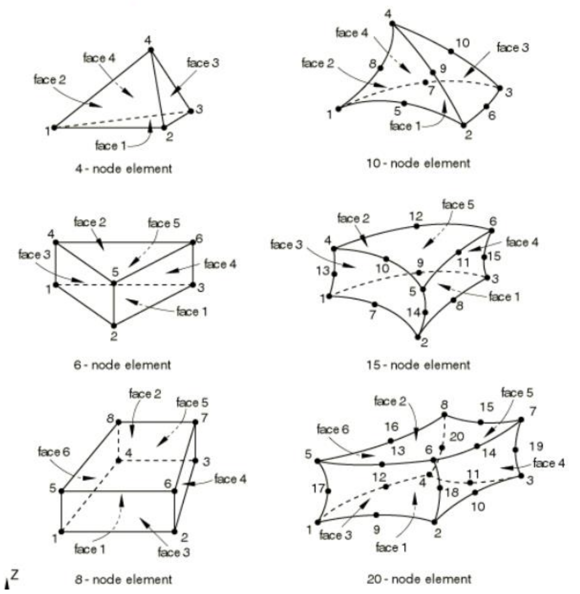
| 1 | **Aspect of the User Case/System to be Simulated** | |
|---|---|---|
| 1.1 | **Aspect of the User Case to be simulated** | *Calculation ion of robustness in front of external loads and minimizing its weight of support geometry considering performance of materials in service.* |
| 1.2 | **Material** | *Mirror's support: material comes from model 3.* |
| 1.3 | **Geometry** | *To be defined but restricted to planar mirror.* |
| 1.4 | **Time Lapse** | *Quasi-static* |
| 1.5 | **Manufacturing process or in-** | *In service conditions: external loads and own weight.* |

| | | |
|---|---|---|
| | SERVICE CONDITIONS | |
| 1.6 | PUBLICATION | |

| 2 | GENERIC PHYSICS OF THE MODEL EQUATION | | |
|---|---|---|---|
| 2.0 | MODEL TYPE AND NAME | *Continuum model for industrial application. Solid mechanics. (model 4.1 in ROMM)* | |
| 2.1 | MODEL ENTITY | *The entity in this materials model is finite volumes of optimal GRFP composite from model 4.* | |
| 2.2 | MODEL PHYSICS/ CHEMISTRY EQUATION PE'S | **Equations** | $\tau_{xy} = \left[\frac{\partial \omega}{\partial y} - (z - z_C)\right] \cdot \frac{M_x}{J}$<br>$\tau_{xz} = \left[\frac{\partial \omega}{\partial z} - (y - y_C)\right] \cdot \frac{M_x}{J}$ |
| | | **Physical quantities for each equation** | *Where,*<br>$\tau$*: strain*<br>*yc, zc: coordinates of the cutting center section*<br>$\omega$*: warpage of the structure.*<br>*Mx: torque.*<br>*J: torsor module.* |
| | MATERIALS RELATIONS | **Equations** | *For rectangular section:*<br>$J = \frac{1}{3} b^3 h \left[1 - \frac{192b}{h\pi^5} \sum_{k=1,3,\dots}^{\infty} \tanh\left(\frac{kh\pi}{2b}\right)\right]$<br>*For any other section (circular, elliptical):*<br>$J = \int_A \left[(y - y_c)^2 + (z - z_c)^2 - (y - y_c)\frac{\partial \omega}{\partial z} + (z - z_c) + (z - z_c)\frac{\partial \omega}{\partial y}\right] dydz$ |
| | | **Physical quantities/ descriptors for each MR** | *Where,*<br>*b, h: section of the structure where b is higher than h.* |
| 2.4 | SIMULATED INPUT | *Optimal GFRP composite properties from previous model 3* | |
| 2.5 | PUBLICATION | | |

| 3 | SPECIFIC COMPUTATIONAL MODELLING METADATA | |
|---|---|---|
| 3.1 | NUMERICAL SOLVER | *Finite element method* |
| 3.2 | SOFTWARE TOOL | *ABAQUS from SIMULIA Platform* |
| 3.3 | TIME STEP | *During a static step a time period to the analysis can be assigned. This is necessary for cross-references to the amplitude options, which can be used to determine the variation of loads and other externally prescribed parameters during a step. In some cases this time scale is quite real. If a time period is not specified, Abaqus defaults to a time period in which "time" varies from 0.0 to 1.0 over the step. The "time" increments are then simply fractions of the total period of the step.* |

| | | | |
|---|---|---|---|
| 3.4 | COMPUTATIONAL REPRESENTATION *Refers to how your computational solver represents the material,* | PHYSICS EQUATION, MATERIAL RELATIONS, MATERIAL | *Static analysis is used for stable problems and can include linear or nonlinear response. The total stress is defined from the total elastic strain as* $\sigma = D^{el}\varepsilon^{el}$*, where* $\sigma$ *is the total stress,* $D^{el}$ *is the forth-order elasticity tensor, and* $\varepsilon^{el}$ *is the total elastic strain.*<br><br>*For this model different elements can be selected according to the problem requirements in order to minimize the calculation error:* |

| | | | |
|---|---|---|---|
| | properties, equation variables, | | <br>Material relations (for anisotropic characteristics). The stress-strain relations are as follows:<br>$$\begin{Bmatrix}\sigma_{11}\\ \sigma_{22}\\ \sigma_{33}\\ \sigma_{12}\\ \sigma_{13}\\ \sigma_{23}\end{Bmatrix} = \begin{bmatrix} D_{1111} & D_{1122} & D_{1133} & D_{1112} & D_{1113} & D_{1123}\\ & D_{2222} & D_{2233} & D_{2212} & D_{2213} & D_{2223}\\ & & D_{3333} & D_{3312} & D_{3313} & D_{3323}\\ & & & D_{1212} & D_{1213} & D_{1223}\\ & sym & & & D_{1313} & D_{1323}\\ & & & & & D_{2323}\end{bmatrix} \begin{Bmatrix}\varepsilon_{11}\\ \varepsilon_{22}\\ \varepsilon_{33}\\ \gamma_{12}\\ \gamma_{13}\\ \gamma_{23}\end{Bmatrix} = [D^{el}] \begin{Bmatrix}\varepsilon_{11}\\ \varepsilon_{22}\\ \varepsilon_{33}\\ \gamma_{12}\\ \gamma_{13}\\ \gamma_{23}\end{Bmatrix}$$<br>Where the material stiffness parameters ($D_{ijkl}$) are giver directly, Abaqus imposes the constraint $\sigma_{33} = 0$ for the plane stress case to reduce the material's stiffness matrix as required.- Each $D_{ijkl}$ is function of Young's modulus and Poisson's ratio for each direction.<br>In Abaqus spatially varying anisotropic elastic behaviour can be defined for homogeneous solid continuum elements by using a distribution. The distribution must include default values for the elastic moduli. If a distribution is used, no dependences on temperature and/or field variables for the elastic constants can be defined.<br><br>Computational representation of the physics equation, materials relation and material ( e.g. "written up for the entity in the model" or in the case of statistical approached "written up for finite volumes") |
| | | **Boundary Conditions** | The previous system of equations is furthermore subject to essential boundary conditions, e.g. prescribed displacements or velocities along a surface, mechanical conditions, etc. In this software boundary conditions are represented into the simulated geometry. |
| | | **Additional Solver Parameters** | 1. Residual control ($R_n^\alpha$) is the convergence criterion for the ratio of the largest residual to the corresponding average flux norm. The default value is $R_n^\alpha$= 5x10-3, which is rather strict by engineering standards but in all but exceptional cases will guarantee an accurate solution to complex nonlinear problems. The value for this ratio can be increased to a larger number if some accuracy can be sacrificed for computational speed.<br>2. The torque each 0.01 meters structure is evaluated. |
| **3.5** | **Publication** | Not publication | |

| **4** | **POST PROCESSING** |
|---|---|
| **4.1** | **The processed output is** The optimal design for mirror support will be defined in this activity with high robustness in front of external loads and minimizing its weight. Also, its design will be |

<table>
<tr><td></td><td>CALCULATED FOR</td><td>used for define planar mirror in model 1, for empirical validation in pilot CSP plant.</td></tr>
<tr><td>4.2</td><td>METHODOLOGIES</td><td>The postprocessing calculator performs the following calculations on data written to the output database:<br>▪ Extrapolation of integration point quantities to the nodes or interpolation of integration point quantities to the centroid of an element, according to the user-specified position for element output.<br>▪ Calculation of history output at tracer particles<br>By default, the postprocessing calculator will run automatically upon the completion of an analysis. During the execution of the analysis, Abaqus will determine if there are keywords in the input file that require the use of the calculator and will initiate the calculator upon completion if it is required. Additionally, the postprocessing calculator can be run manually.</td></tr>
<tr><td>4.3</td><td>MARGIN OF ERROR</td><td>The ability of a finite element analysis to make useful predictions of physical behavior depends on many factors, including:<br>- representation of geometry, material behavior, load history, and various other modeling aspects associated with describing the problem posed;<br>- spatial and temporal discretization (mesh refinement and incrementation); and<br>- convergence tolerances.<br>The finite element discretization of a model domain results in an approximation to the exact solution for all but trivial analyses. For aiding in understanding the extent and spatial distribution of the discretization error in a finite element solution, Abaqus provides a set of error indicator output variables. Ideally, error indicator output variables should be supplemented by other techniques, such as a mesh refinement study, to gain confidence that discretization error is not significantly degrading the ability of the finite element analysis to make useful predictions. In fact, error indicators can help automate a mesh refinement study through the adaptive remeshing functionality of Abaqus/CAE.<br>Abaqus error indicator variables provide a measure of the local error resulting from mesh discretization. Each error indicator, provides an indication of error in a particular base solution variable. Following table shows the available error indicator variables and the corresponding base solution variables.
<table>
<tr><th>Solution Quantity</th></tr>
<tr><td>Element energy density</td></tr>
<tr><td>Mises stress</td></tr>
<tr><td>Contact pressure</td></tr>
<tr><td>Contact shear stress</td></tr>
<tr><td>Equivalent plastic strain</td></tr>
<tr><td>Plastic strain</td></tr>
<tr><td>Creep strain</td></tr>
<tr><td>Heat flux</td></tr>
<tr><td>Electric flux</td></tr>
<tr><td>Electric potential gradient</td></tr>
</table>
The algorithms used by Abaqus/CAE to modify mesh seed sizes for the adaptive remeshing capability consider error indicator values and corresponding base solution values together. When a remeshing rule is created and requested a particular error indicator, Abaqus automatically writes the error indicator and corresponding base solution variable to the output database.<br><br>The error indicators are approximate measures of the local error in the base solution and are, themselves, subject to discretization error. The accuracy of the error estimates tends to improve as the mesh is refined. Each error indicator variable has the same units has the corresponding base solution variable, which facilitates comparison of local estimates of the error magnitude with local estimates of the base solution.</td></tr>
</table>