Structural Mechanics of Steam Turbines: Facing challenges in FE-postprocessing with STARpost

Dr. Benjamin Kloss-Grote

Siemens AG, Energy Sector
Fossil Power Generation
Rheinstr. 100, 45478 Muelheim, Germany

Abstract: In isothermal machine parts, the location of the highest stress can often be regarded as the critical location regarding the structural integrity. Steam turbines, like many other products in hot turbomachinery possess an inhomogeneous temperature distribution during operation. As the relevant material properties are temperature dependent, the stress level is no longer a sufficient indicator of the mechanical integrity: Low stress levels in high temperature regions can lead to the same or a higher mechanical utilization than high stress levels in colder regions. This effect poses a challenge for FE post processing because temperatures, material properties and stress levels need to be considered together for an assessment of the mechanical integrity. Abaqus offers the user subroutine UVARM to calculate user defined variables that can meet this challenge, but, especially for long lasting creep calculations, a pure post processing approach is much more convenient.

This paper describes how Siemens Energy large scale steam turbines R&D has faced this challenge by developing 'STARpost' (i.e. Steam Turbine Analysis Routine – postprocessing), an efficient post-processing-plug-in for Abaqus/CAE: 'STARpost' automatically reads in the relevant temperature-dependent material data, assesses the existing stresses against the varying allowable stresses (for components with different materials and different calculation steps), provides many more field outputs beneficial for hot turbomachinery R&D and an environment for convenient visualization.

Keywords: Creep, Elasticity, Failure, Heat Transfer, Mechanical Integrity, Output Database, Plasticity, Postprocessing, Python Scripting, Plug-In, Strength Assessment, Thermal Stress, Turbines, Visualization.

1. Introduction

1.1 Steam Turbines

Large scale Siemens Steam Turbines are installed in steam power plants, combined cycle power plants and nuclear power plants with a power output between 90 MW and 1900 MW per turboset (example see Fig. 1).
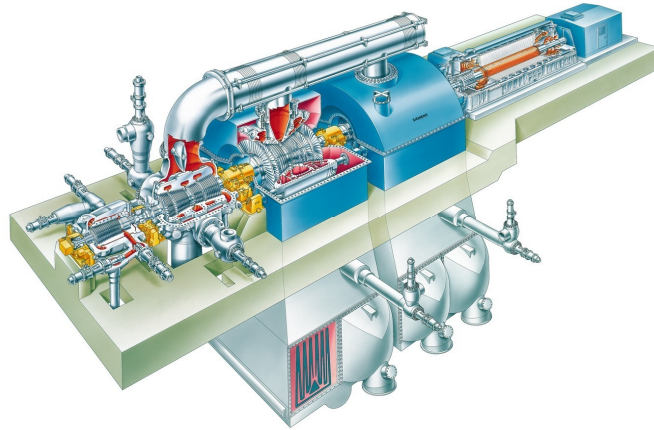
Figure 1. Siemens Steam Turbine SST5-6000 (Power Output 600 MW; 1 high pressure steam turbine, 1 intermediate pressure steam turbine, 2 low pressure steam turbines and 1 generator). Image courtesy of Siemens AG.

An essential part in the development of new steam turbines is to ensure the mechanical integrity of all components. Steam turbines have to withstand a variety of transient and steady-state loads which can be divided into primary (i.e. force/pressure based, centrifugal or gravity loads) and secondary (thermal based; self equilibrating) loads. With main steam parameters of up to 600°C and 300 bar, reheat steam temperatures of up to 620°C (Siemens AG Energy Sector, 2010) and an operational lifetime of over 25 years, creep effects have to be taken into account for the long term behavior of these components. Due to the complexity of geometries and loads, 2D and 3D finite element analyses are applied for for example heat transfer, static and creep analyses.

This paper is focused on the mechanical integrity of large components like the casings and the rotors of high and intermediate pressure steam turbines (also called 'hot' steam turbines, i.e. operating at elevated temperatures) under static loads and long-term steady-state loads including creep behavior.

1.2   Standard load cases in scope of this paper

Standard load cases to analyze the static and creep behavior of the large 'hot' steam turbine components with Abaqus/Standard are for example

A.   Heat Transfer: The steady state temperature distribution in the assembly is derived by a heat transfer analysis with a selection of surface film interactions, thermal conductance contact interactions, radiation interactions and temperature boundary conditions, see (Abaqus/CAE User's Manual, 2010).

B.   Static Primary elastic: For this elastic analysis step, the steady state temperature distribution from the heat transfer analysis is read into the structural analysis which contains all mechanical loads (steam pressures, forces, moments etc.). As no secondary (i.e. thermal based) loads are taken into account, no thermal expansion is considered in the material models for this step.

C. Static Primary+Secondary elastic: The same conditions as for B apply – now including the thermal expansion in the material model.

D.1 Static Primary+Secondary elastic-plastic: The same conditions as for C apply – now including elastic-plastic material behavior.

D.2 Creep Primary+Secondary elastic-plastic-creep: The creep step follows step D.1 with time points or intermediate steps at interesting times (for example maintenance intervals). The total creep time is the required operational lifetime of the steam turbine.

1.3 Creep basics

(Naumenko and Altenbach, 2007) define *creep* in the context of uniaxial homogenous stress states in material testing as 'the progressive time-dependent deformation under constant load and temperature'. For *creep in structures*, they offer a broader definition: i.e. 'time-dependent changes of strain and stress taking place in structural components as a consequence of external loading and temperature'. For further general creep basics see for example (Betten, 2008).

For the strength assessment of mechanical designs, two temperature ranges have to be considered: In the 'non creep range', the nominal design stresses can be determined using time independent material characteristics. In the (high temperature) creep range, this is no longer the case (DIN EN 13445-3, 2010) and the creep behavior has to be taken into account. These two ranges can be separated by the intersection temperature T_i – at the intersection of the yield strength $R_{p0.2,T}$ and the creep rupture strength for 1e5h of creep $R_{m,1e5h,T}$ (see Figure 2):
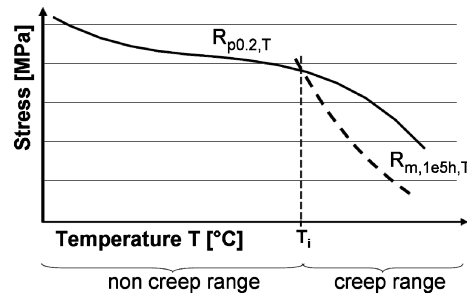

Figure 2. Determination of the creep range with the intersection temperature T_i (Example: high chromium cast steel; explanations in the text).

Several approaches exist for modeling the viscoplastic material behavior during creep. Depending on the problem and the required accuracy of the results, different approaches are used in steam turbine structural mechanics: A simple approach is the Norton-Bailey-law (Bailey, 1930; Norton, 1929) for (secondary) steady-state creep (Equation 0, sometimes called power law creep law) in its time-hardening formulation

$$\varepsilon^{cr} = A_1 \cdot \sigma^{A_2} \cdot t^{A_3},$$ (0)

with the (uniaxial) creep strain ε^{cr}, stress σ, creep time t and the three temperature dependent, material specific creep constants A_1, A_2 and A_3 (Naumenko and Altenbach, 2007). More advanced approaches include for example the Garofalo creep law (Garofalo, 1963) and many others (see for example (Gorash, 2008)).

1.4 Terminology for the strength assessment

In a standard assessment of the mechanical integrity of an assembly, existing stresses σ (or existing strains) are assessed against allowable stresses (or allowable strains)

$$\sigma = \sigma_{existing} \overset{!}{\leq} \sigma_{allowable} = \frac{\text{Material Strength}}{SF_{allowable}} \, , \qquad (1)$$

whereby the allowable stresses can be derived from the material strength (for example yield strength $R_{p0.2,T}$, ultimate tensile strength $R_{m,T}$, creep rupture strength $R_{m,t,T}$ etc., with temperature T and creep time t), divided by the allowable safety factor $SF_{allowable}$ (see Equation 1). The allowable safety factor, sometimes called required safety factor, design safety factor or design factor, is in general specified for different stress categories like membrane or bending stresses by experience, company regulations or external standards. To confirm the mechanical integrity, the inequality relation in Equation 1 has to be fulfilled.

Often, it is not only necessary to confirm Equation 1 to be true, but to determine the degree in which the existing stress in the assessed design 'uses up' the allowable stress. In this paper, this metric is called (existing) *safety utilization SU* and defined as (Equation 2)

$$SU = SU_{existing} = \frac{\sigma_{existing}}{\sigma_{allowable}} = \frac{\sigma_{existing} \cdot SF_{allowable}}{\text{Material Strength}} \, . \qquad (2)$$

The safety utilization has the dimension of 1. A safety utilization of 0% implies that there is no load present. The mechanical integrity according to Equation 1 is confirmed if the safety utilization shows values between 0% and 100%. For safety utilizations greater than 100%, the structure fails under the given circumstances.

With the definition of the *existing safety factor $SF_{existing}$*, (sometimes also called *realized safety factor*) (Equation 3)

$$SF_{existing} = \frac{\text{Material Strength}}{\sigma_{existing}} \, , \qquad (3)$$

the strength assessment in Equation 1 can be written as (Equation 4)

$$SF_{existing} \overset{!}{\geq} SF_{allowable} \, . \qquad (4)$$

(Be aware of the changed inequality sign in comparison to Equation 1.)
Especially for non-isothermal FE post processing, it is very effective to define the reciprocal value of the existing safety factor

$$MU = MU_{existing} = \frac{1}{SF_{existing}} = \frac{\sigma_{existing}}{\text{Material Strength}} \qquad (5)$$

and call it (existing) *material utilization MU*. Safety utilization and material utilization depend on each other according to Equation 6, derived from Equation 5 and 2:

$$SU = MU \cdot SF_{allowable} \, , \tag{6}$$

To derive allowable values for the safety utilization, the remarks below Equation 2 lead to Equation 7:

$$SU_{allowable} = 100\% \, . \tag{7}$$

The *allowable material utilization* can be derived from Equation 7 and 6 (Equation 8):

$$MU_{allowable} = \frac{1}{SF_{allowable}} \, , \tag{8}$$

The strength assessment Equation 1, written for the safety utilization, reads (Equation 9)

$$SU_{existing} \overset{!}{\leq} SU_{allowable} = 100\% \, , \tag{9}$$

and for the material utilization (Equation 10)

$$MU_{existing} \overset{!}{\leq} MU_{allowable} = \frac{1}{SF_{allowable}} \, . \tag{10}$$

The respective definitions have been added in Equations 11 and 12 in order to provide a two-equation-summary of the utilization-based strength assessments:

$$SU_{existing} = \frac{\sigma_{existing} \cdot SF_{allowable}}{\text{Material Strength}} \overset{!}{\leq} SU_{allowable} = 100\% \, , \tag{11}$$

$$MU_{existing} = \frac{\sigma_{existing}}{\text{Material Strength}} \overset{!}{\leq} MU_{allowable} = \frac{1}{SF_{allowable}} \, . \tag{12}$$

For the strength assessment of isothermal assemblies, Equations 9 to 12 (i.e. the utilization based formulation) do not present any significant advantage in comparison to Equation 1 (the stress based formulation) other than a scaling of the results, because the material strength is a constant for one material over the whole assembly. The great advantages for non-isothermal assemblies in FE post processing will be presented in Chapter 2.

2. The Challenge

The structural analyst who wants to perform strength assessments for steam turbines at elevated temperatures (for example with load cases according to Chapter 1.2) faces several challenges during his or her FE analysis:

The illustration example selected for this paper represents the upper part of an inner casing of a Siemens intermediate pressure (IP) steam turbine with operational temperatures below and within the creep range (the blading is not part of this analysis).

2.1 Inhomogeneous temperature distribution prevents stress based assessments

For an isothermal assembly, the (existing) stress results can be assessed against one single value of the allowable stress (Equation 13, with the assumption of a constant safety factor):

$$\sigma = \sigma_{existing}(x,y,z) \overset{!}{\leq} \sigma_{allowable} = \frac{\text{Material Strength}}{SF_{allowable}} = const\,, \qquad (13)$$

This can be easily performed in Abaqus/CAE during post processing for example by setting the upper limit of the contour plot options to the allowable stress and/or using isosurface view cuts (see (Abaqus/CAE User's Manual, 2010)): Grey/displayed areas exceed the allowable stress limit. Critical locations can be therefore easily identified.

For assemblies with a non-uniform temperature distribution, the allowable stress varies over the whole assembly with the temperature, because the material strength varies with the temperature.

$$\sigma = \sigma_{existing}(x,y,z) \overset{!}{\leq} \sigma_{allowable}(x,y,z) = \frac{\text{Material Strength}(x,y,z)}{SF_{allowable}}\,, \qquad (13)$$

In Figure 3, the contour plot of the equivalent stress (von Mises stress) is shown for the illustrated example with an inner casing of an intermediate pressure (IP) Siemens Steam Turbine.
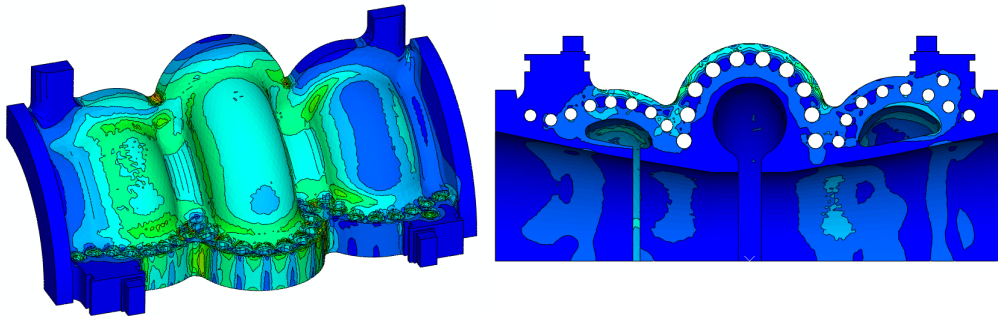

Figure 3. Equivalent stress distribution during steady-state operation in the upper part of the inner casing of an intermediate pressure (IP) steam turbine [left: outside view, right: inside view].

Inside the turbine at the bladepath, the equivalent stresses remain almost constant.

The steady-state temperature distribution (NT11) of the assembly, derived from the heat transfer analysis, is pictured in Figure 4: This temperature distribution is clearly non-isothermal, so the material strength of Equation 13 varies spatially according to Figure 2 and Figure 4.
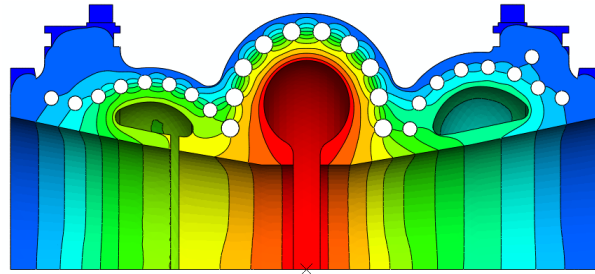
Figure 4. Non-isothermal temperature distribution in steady-state operation of the inner casing of an IP steam turbine.

Therefore, the stress level is no longer a sufficient indicator of the mechanical integrity: Low stress levels in high temperature regions can lead to the same or a higher mechanical utilization than high stress levels in colder regions. In addition, stress redistribution may also change the mechanical utilization along a critical cross section.

The only effective way to assess the mechanical integrity of these assemblies is to apply a procedure that automatically calculates the temperature dependent allowable stress (or material strength) for every node (or integration point) of the assembly and assesses the existing stress against it. Here, the definitions of the safety utilization SU or the material utilization MU (see Chapter 1.4) come in very handy for a strength assessment (Equation 14 and 15)

$$SU_{existing}(x,y,z) = \frac{\sigma_{existing}(x,y,z) \cdot SF_{allowable}}{\text{Material Strength}(x,y,z)} \overset{!}{\leq} SU_{allowable} = 100\% , \qquad (14)$$

$$MU_{existing}(x,y,z) = \frac{\sigma_{existing}(x,y,z)}{\text{Material Strength}(x,y,z)} \overset{!}{\leq} MU_{allowable} = \frac{1}{SF_{allowable}} , \qquad (15)$$

because the spatial dependency is contained in one single expression on the left side of the equation. The allowable value on the right side has become a constant in this formulation (assumption for Equation 14 and 15: $SF_{allowable}$=const). This allows for easy visualization of critical locations by using the limits in contour plot options and isosurface view cuts.

To conclude it can be stated that for strength assessments of non-isothermal assemblies, new field outputs need to be created, the SU or MU.

2.2 Material strength is not contained in inp and ODB

For strength assessments in the creep range, one relevant material strength is the creep rupture strength $R_{m,t,T}$. This material property is not part of the common material models in Abaqus. Therefore it cannot be included in the input file and it is subsequently not available for the solver and finally not in the ODB for post processing.

Hence the creep rupture strength needs to be made available to the algorithm that calculates the SU or the MU by a different way than the input file. (This applies as well for rupture strains in strain assessments, which have not been mentioned before.)

2.3 Assemblies with more than one material

If more than one material is used in the assembly, the complexity increases. The calculation of the SU or the MU requires the knowledge of the material association for each element. Simple post processing approaches could involve the use of display groups for each material. This approach a) becomes very tedious with an increasing number of materials. In addition, it is b) always a possible quality threat for the assessment if the assembly is not evaluated with a complete overview but only in individual pieces of the whole puzzle: There is the risk that important effects at the intersection of two materials can be overlooked or not properly understood.

2.4 Many frames need to be evaluated

It can be concluded from Chapter 1.2, that the strength assessment of 'hot' steam turbines requires taking many analysis frames into account. An automatic evaluation procedure has to support that.

2.5 Many different output variables necessary

Many different stress outputs (for example equivalent stress, principal stresses, shear stresses etc.) need to be able to be taken into account – depending on the expected failure mechanism. An automatic evaluation procedure has to support that.

2.6 Varying allowable safety factors

Standards relevant for the mechanical design of steam turbines provide different safety factors for different materials (e.g. ductile vs. brittle), for different material models (e.g. elastic vs. elastic-plastic), for different load cases (e.g. primary vs. primary+secondary loads), for different locations in an assembly (e.g. local vs. global location), for different temperatures in an assembly (e.g. creep vs. no creep range), for different categories of linearized stresses (e.g. membrane vs. bending vs. peak stresses) etc.

Therefore, not only the existing stresses and material strength vary spatially, but also the allowable safety factor. The utilization based strength assessment equations with varying safety factors look like this (Equations 16 and 17):

$$SU_{existing}(x,y,z) = \frac{\sigma_{existing}(x,y,z) \cdot SF_{allowable}(x,y,z)}{\text{Material Strength}(x,y,z)} \overset{!}{\leq} SU_{allowable} = 100\%, \quad (16)$$

$$MU_{existing}(x,y,z) = \frac{\sigma_{existing}(x,y,z)}{\text{Material Strength}(x,y,z)}$$

$$\overset{!}{\leq} MU_{allowable}(x,y,z) = \frac{1}{SF_{allowable}(x,y,z)}, \quad (17)$$

It is obvious at first glance, that with varying safety factors, an effective strength assessment would require the use of the SU formulation (Equation 16), because the right side of the equation is still a constant. The MU formulation (Equation 17) would still need to work with display groups or other different views.

3. Existing solutions in Abaqus

The user subroutine UVARM (Abaqus User Subroutines Reference Manual, 2010) is provided for Abaqus/Standard to generate element output. With UVARM, the safety utilizations and material utilizations can be calculated during the solver process. The limitations of the application of UVARM in regard to the challenges mentioned in Chapter 2 are: a) no self-explanatory output variable names can be used, which can lead to significant confusion and quality issues, b) Calculation of user output variables with UVARM require a solver run. If a user output has not been requested before the solver run and should be added afterwards to the ODB – a new solver run is required. This is especially inconvenient for long lasting creep calculations that may take up to several weeks. c) Many material properties need to be hard coded into the subroutine, which is especially tedious when using many materials and if material properties are updated.
To conclude it can be stated that it is possible to calculate the utilizations with UVARM, although the limitations regarding effectiveness and possible quality concerns apply.

4. Siemens Energy's solution: STARpost

4.1   Introduction

To face the challenges mentioned in Chapter 2 and to compensate the limitations of a strength assessment based on UVARM mentioned in Chapter 3, Siemens Energy large scale steam turbine R&D have developed a customized strength assessment post processing approach as Python Plug-Ins for the visualization module of Abaqus/CAE: STARpost.
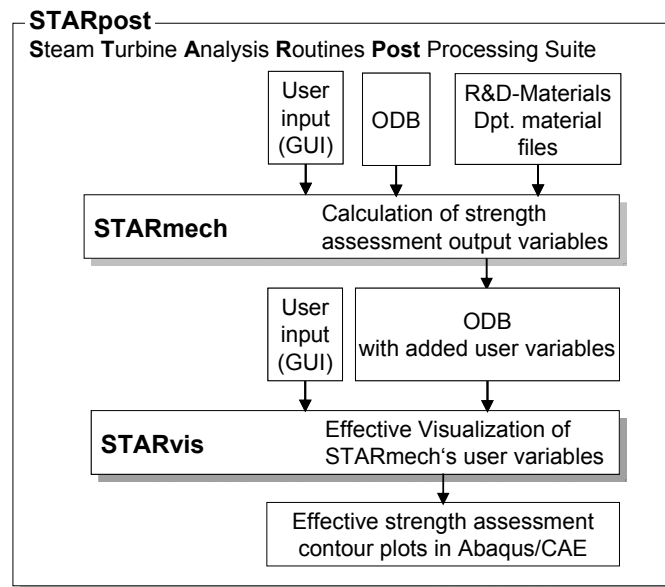

Figure 5. Structure of STARpost, the post processing suite for static and creep analyses, including relevant inputs and outputs.

As can be seen in Figure 5, STARpost consists of two modules: STARmech, the plug-in for the calculation of strength assessment output variables, and STARvis, the plug-in for effective visualization of STARmech's user variables:

The workflow of a strength assessment with STARpost (including Figure 5) will be described on the basis of the respective GUIs in the following sections.

4.2   Application of STARmech for the calculation of the strength assessment output variables

After a static/creep run of the Abaqus/Standard solver, the ODB without any user defined output variables is loaded (in writeable mode; automatically or manually) into STARmech, the plug-in for the calculation of strength assessment output variables. See Figure 6 for a screenshot of the STARmech GUI. The following numbers in brackets (..) refer to Figure 6. The ODB-file name is displayed under (1.), the job description can be found under (2.) to confirm that the correct ODB has been selected.
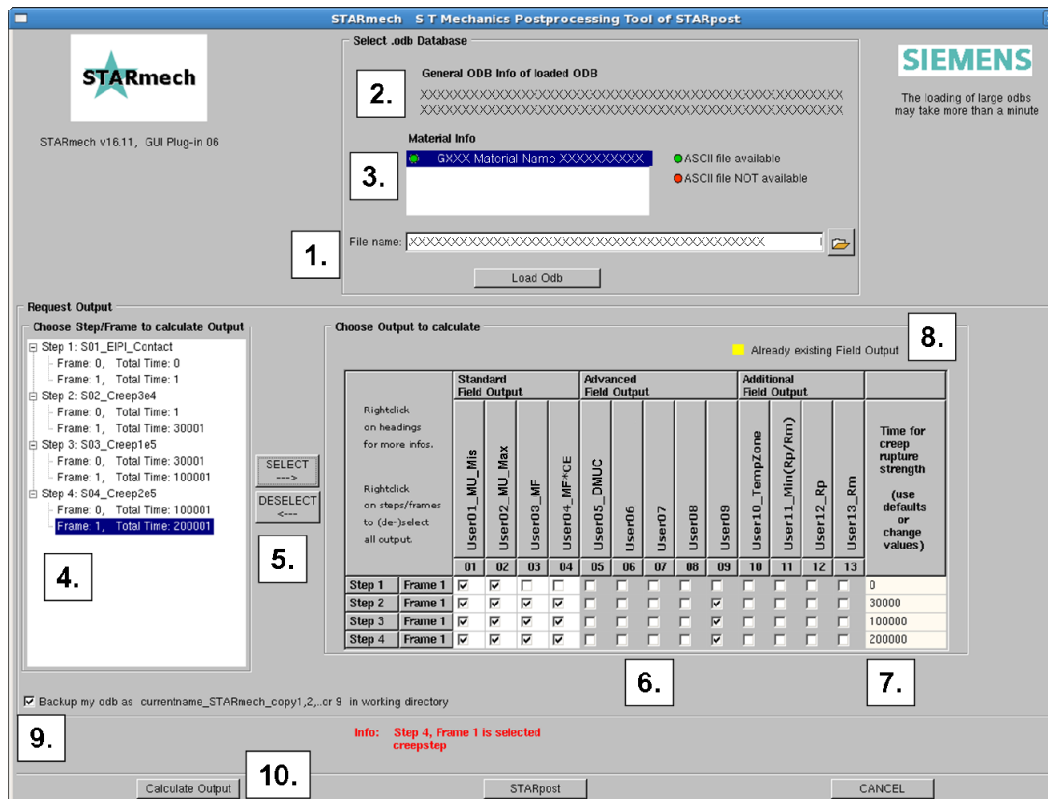

Figure 6. Screenshot of the GUI of the STARmech-Plug-In: requesting user field outputs for an efficient strength assessment during post processing - suitable for non-isothermal, multi-material, multi-frame static and creep analyses (The numbers are explained in the text).

STARmech scans the ODB for all used materials and displays them under (3.). The screenshot shows a very basic static and creep ODB for the inner casing illustration example with one single material. The green traffic light (3.) indicates that the correct ASCII material file for the material used in the ODB has been found in a standard subfolder of the working directory. [The material import during pre-processing, which is handled by a separate plug-in, does not only import all Abaqus-relevant material data from a standard ASCII material file from the R&D materials department into the MDB, but also copies the used ASCII material file (including time, date etc. of the import) into a subfolder. This storage procedure ensures that the correct revision of the correct material data is associated with the ODB. For post processing, all material properties that cannot be contained in the input file can be accessed very easily via this ASCII file.]

All steps and frames are displayed under (4.), wherefrom the steps of interest can be selected by (5.) for output variable requests into the table (6.). STARmech automatically detects creep steps and proposes a different set of default output variables for static and creep steps. The creep time of the selected step (not the total time) is set as the default value for the creep rupture time, but can be altered by the user (7.). If the loaded ODB would already contain STARpost user defined variables from a previous assessment, these variables would be highlighted in yellow (8.). This is important because field output variables in the ODB cannot be deleted due to requirements by Abaqus.

As a default, a backup of the loaded ODB is made (9.) before the calculation of the variables starts. When the calculation is initiated by pressing (10.), the GUI sends the respective command to the kernel including all user input data. All relevant process steps of the STARmech kernel are displayed in the message window of Abaqus/CAE and logged in a logfile for reasons of quality assurance and debugging in case of error messages. The kernel finishes its work with displaying the 'new' ODB with the added field outputs in write-protected mode (see also Figure 5) and switches to the STARpost-GUI:

4.3   Application of STARvis for effective visualization of the strength assessment output variables

In Figure 7, the GUI of STARpost is presented. STARpost has been developed as well as a Python plug-in available in the Visualization Module of Abaqus/CAE. The following numbers in brackets (..) refer to Figure 7. STARmech can be started under (1.). The area below (1.) belongs to STARvis: The steps and frames of the ODB currently displayed in the active viewport are listed under (2.). If a frame is selected, the user defined field outputs that have been calculated by STARmech within this frame are displayed under (3.). If a field output is selected, the viewport is automatically updated with the contour plot of this field output. The most important plot options can be set under (4.).

All user field outputs generated by STARmech are regular user field outputs and can be accessed as well by the standard ways in Abaqus/CAE. The advantage of using the STARpost-GUI for visualization is that for every STARpost user field output, the most advisable, specific set of plot/legend/… options is stored. This allows for standardized post processing on one hand and for specific color coding of selected field outputs on the other.
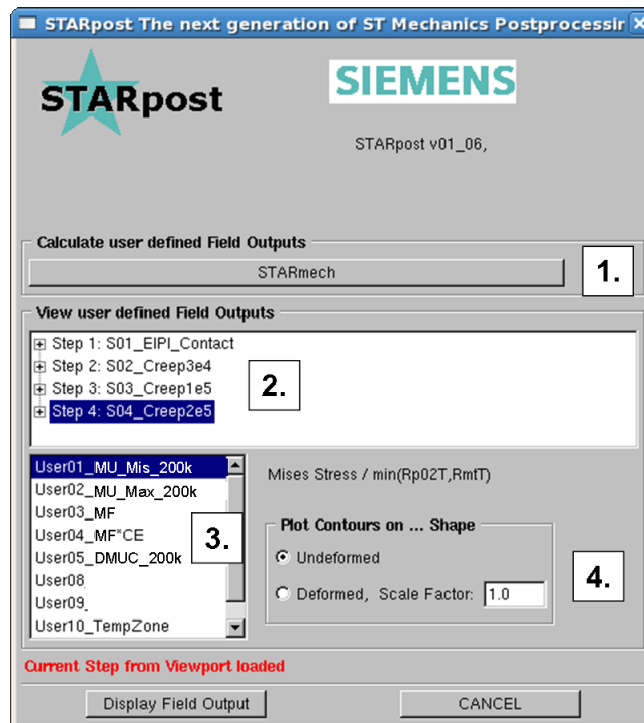
Figure 7. Screenshot of the GUI of the STARpost-Plug-In: effective visualization of the user output variables calculated by STARmech (The numbers are explained in the text).

4.4   STARpost user variables

The STARpost user field outputs can be clustered into three groups: 1. 'Standard' user field outputs that are essentially required for a static/creep strength assessment of non-isothermal assemblies, 2. 'Advanced' user field outputs that foster plausibility checks and can provide deeper insights into the mechanical mechanisms within the assembly and 3. 'Research' user field outputs that are only required for very deep understanding in high-innovation projects. They are not explained in this paper.

4.4.1   Standard STARpost user field outputs

Central user field output is the material utilization MU (see Equation 5), based on Mises (equivalent) stresses for ductile materials and ductile stress states (User01) and User02, based on the maximum of Mises and absolute Principal Stresses for more brittle materials and brittle stress states. The safety utilization is not implemented into STARpost due to the vast number of different allowable safety factors (see section 2.6).

To enable also a strain based strength assessment, the weighted creep strains can be calculated with User04. The multi-axiality-factor MF (User03) accounts for effects that cannot be modeled properly with an uniaxial creep law like Norton-Bailey.

Table 1. Standard STARpost strength assessment user field outputs.

Name	Description
User01_MU_Mis	Material Utilization with Mises (eq.) Stress $= \dfrac{\sigma_{Mises}}{\min(R_{m,t,T}, R_{p,T}}$
User02_MU_Max	Material Utilization with maximum Stress $= \dfrac{\max(\sigma_{Mises}, \lvert\sigma_I\rvert, \lvert\sigma_{III}\rvert)}{\min(R_{m,t,T}, R_{p,T})}$ }
User03_MF	Multi-axiality-factor (MF) $= f(\sigma_{hydro}, \sigma_{Mises}, Material\ Constant)$
User04_MF*CE	Multi-axiality-factor * Max. Principal Creep Strain $= MF \cdot \varepsilon_I^{cr}$

A selection of the STARpost user variables is presented in the illustrated example: The Mises material utilization (User01) is presented in Figure 8. Inside the turbine, the Mises MU differs significantly from the Mises stress plot of Figure 3.
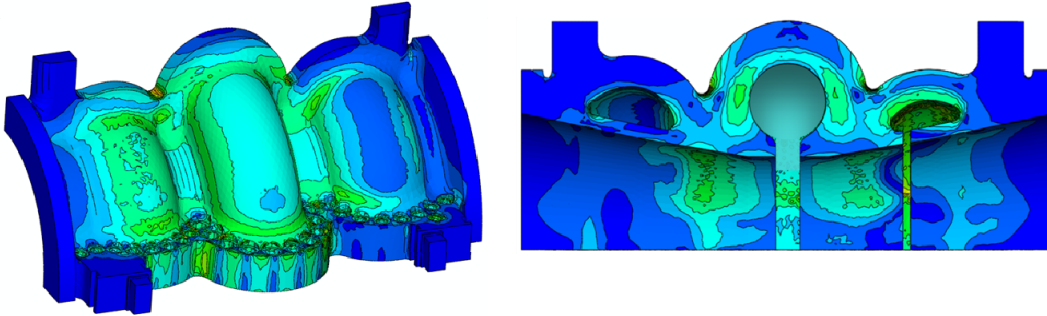

Figure 8. STARpost user output variables: Mises material utilization (User01).

User02 is not presented in a Figure. The multi-axiality-factor MF shows only some slight peaks at the vertical rib (see Figure 9, left). The contour plot of the (max. principal creep strain weighted with MF) can be seen in Figure 9, right):
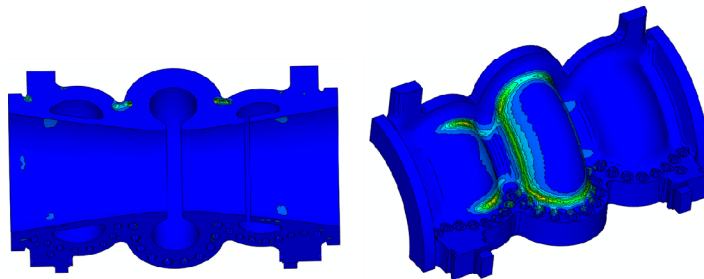

Figure 9. STARpost user output variables: Multi-Axiality-Factor (MF, User03) [left] and MF*max. principal creep strain (User04) [right].

4.4.2   Advanced STARpost user field outputs

Three important temperature zones can be indicated by 'TempZone': the non creep range, the creep range and, if the allowed material temperature is exceeded, the prohibited range (User10). The imported material data can be displayed as well (User11-13). A new user field output is the dominant material utilization criterion (DMUC, User05). This field output indicates which of the three material utilizations of User02_MU_Max dominates in which area of the assembly. The DMUC is very powerful for plausibility checks regarding the loads.

Table 2. Advanced STARpost strength assessment user field outputs.

Name	Description
User10_TempZone	$$\text{Temperature Zone} \quad = \begin{cases} \text{red, } \textit{if} \text{ prohibited range } (T_{allowed} < T) \\ \text{yellow, } \textit{if} \text{ creep range } (T_i \leq T \leq T_{allowed} \\ \text{blue, } \textit{if} \text{ non creep range } (T < T_i) \end{cases}$$ T_i: Intersection Temperature, see Figure 2 $T_{allowed}$: Maximum allowed application temperature of the material
User11_Min(Rp/Rm)	min(Yield Strength $R_{p0.2,T}$, Creep Rupture Strength $R_{m,t,T}$)
User12_Rp	Temperature dependent distribution of Yield Strength $R_{p0.2,T}$
User13_Rm	Temperature dependent distribution of Creep Rupture Strength $R_{m,t,T}$
User05_DMUC	Dominant Material Utilization Criterion of User02_MU_Max $$= \begin{cases} \text{yellow } (\textit{Mises}), \textit{if } MU\_max = \dfrac{\sigma_{Mises}}{\min(R_{m,t,T}, R_{p,T})} \\[2ex] \text{red } (\sigma_I), \textit{if } MU\_max = \dfrac{|\sigma_I|}{\min(R_{m,t,T}, R_{p,T})} \\[2ex] \text{blue } (\sigma_{III}), \textit{if } MU\_max = \dfrac{|\sigma_{III}|}{\min(R_{m,t,T}, R_{p,T})} \end{cases}$$

The non creep range and the creep range can be easily identified by TempZone. No red color indicates that the maximum allowable temperature is not exceeded (see Figure 10).
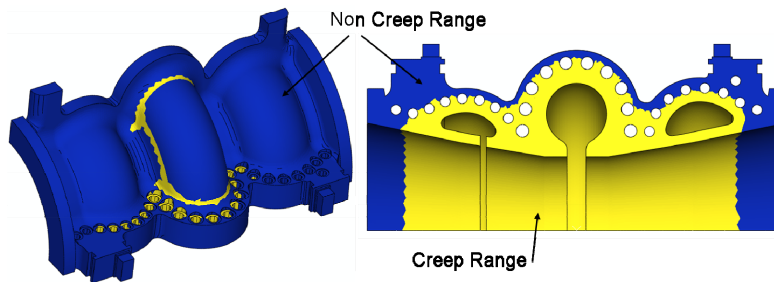

Figure 10. STARpost user field output Tempzone (User10).

Please note that for the display of the material strength (User11, see Figure 11, left), the spectrum has been inverted: red colors indicate a low material strength. The DMUC provides a plausible picture of the inner casing mechanics (User05, Figure 11, right): the inside of the turbine is warmer than the outside and higher pressurized. Thus, tension is expected to dominate on the outside of the turbine (red color, max. principal MU dominates) and compression is expected to dominate on the inside of the turbine (blue, min. principal MU dominates). In the passage between these two areas, the Mises material utilization dominates.
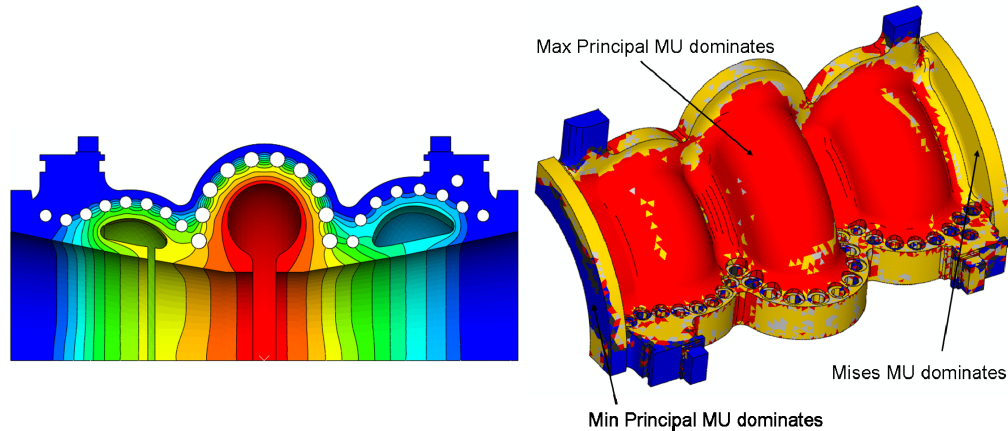

Figure 11. STARpost user field outputs: Material strength – Minimum of Yield Stress and Creep Rupture Stress (User11) [left] and Dominant Material Utilization Criterion (DMUC, User05) [right].

4.5   Summary

Siemens Energy large scale steam turbine R&D have developed a Python based post processing suite for Abaqus/CAE that can effectively generate and visualize strength assessment field output variables which are essential for non-isothermal assemblies that are often part of hot turbomachinery.

5.   References

1. Abaqus User Subroutines Reference Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI, USA, 2010.
2. Abaqus/CAE User's Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI, USA, 2010.
3. Bailey, R. W., "Creep of Steel under Simple and Compound Stress." *Engineering*, 121, 129-265, 1930.
4. Betten, J., Creep Mechanics, Springer, Berlin, Germany, 2008.
5. DIN EN 13445-3, "Unfired Pressure Vessels – Part 3: Design." Beuth Verlag, Berlin, Germany, 2010.

6. Garofalo, F., "An Empirical Relation Defining the Stress Dependence of Minimum Creep Rate in Metals." *Metal Transactions. Trans. AIME*, 227, 351-356, 1963.

7. Gorash, Y., "Development of a Creep-Damage Model for Non-Isothermal Long-Term Strength Analysis of High-Temperature Components Operating in a Wide Stress Range," PhD-Thesis, Martin-Luther-Universität Halle-Wittenberg, Halle an der Saale, Germany, 2008.

8. Naumenko, K., and Altenbach, H., Modeling of Creep for Structural Analysis, Springer, Berlin, Heidelberg, Germany, 2007.

9. Norton, F. H., The Creep of Steel at High Temperature, McGraw-Hill Book Co., New York, 1929.

10. Siemens AG Energy Sector, Siemens Steam Turbines - Large Scale Product Line, Fossil Power Generation Division - Order No. E50001-G210-A137-X-4A00, Erlangen, Germany, 2010.

6. Acknowledgment