ISBN: 978-3-95450-207-3 doi:10.18429/JACoW-MEDSI2018-WEPH07

Content from this work may be used under the terms of the CC BY 3.0 licence (© 2018). Any distribution of this work must maintain attribution to the author(s), title of the work, publisher, and DOI.

PHOTON BEAM APPLIED AS HEAT FLUX ON IRREGULAR SURFACES IN FEA*

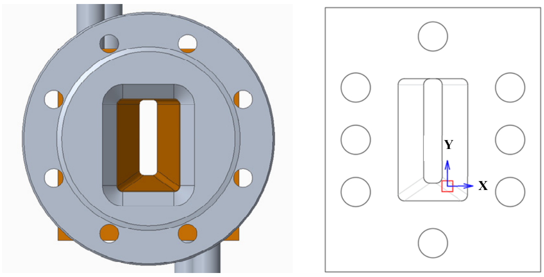
D. Capatina†, Advanced Photon Source, Argonne National Laboratory, Lemont, USA

Abstract

The light source front ends and beamlines contain several devices designed to limit the size of, or completely stop, the photon beam. Most of these devices are meant to protect personnel and/or equipment, thus their failure would have serious implications for the facility operation. The photon beam carries extremely high energy, thus the system will experience very large thermal loads. Accurate temperature and stress distribution of these components, based on well-reasoned assumptions, is needed to accurately review the performance of these devices during the design process. Applying non-uniform heat flux as a thermal load in simulation presents a challenge. This work describes the steps of the thermomechanical numerical simulation for a typical component at the Advanced Photon Source (APS), subject to photon beam interception. The numerical algorithm used to apply the non-uniform heat flux distribution on an irregular type of surface is presented in detail. The algorithm was developed using the commercial Finite Element Analysis (FEA) software ANSYS Workbench of ANSYS, Inc.

PROBLEM FORMULATION

A photon mask (here after mask) is a front end or beamline component meant to limit the size of the photon beam presented to the downstream components. During normal operation the photon beam passes through the mask aperture. By several reasons the beam could be missteered from its normal trajectory, causing the beam to partially or fully strike the mask [1]. Accurate thermal simulation of this mask is needed to review the performance of this device under the worst case load and beam missteering scenarios.



(a) (b)

Figure 1: (a) CAD model of the mask assembly, front view; (b) CAD drawing of the mask model used in the analysis, front view, showing the beam footprint.

A typical mask design is the boxed-cone and the aperture of such component is shown in Fig. 1a. One of the most critical scenarios is when the beam strikes the corner of the taper. The beam footprint for this case is shown in Fig. 1b.

The power density distribution corresponding to the normal beam incidence assuming a 4th order Gaussian equation was calculated using SRUFF [2]. Equation (1), generated for a particular loading scenario for demonstration purposes, was used in ANSYS to apply the heat flux thermal load:

$$q = \exp(a+bx^2+cy^2+dx^4+fy^4+gx^2y^2) \qquad (1)$$

where a=4.3828, b=-0.037375, c=-0.20196, d=-0.0023109, f=0.0045228 and g=-0.020031.

Equation (1) represents the coordinate-dependent power density, q, along the beam axis, z. The power absorbed by the mask surface and thus to be applied in ANSYS is the component of the q vector perpendicular to the surface, q_n. The heat flux vector decomposition is illustrated in Fig. 2.

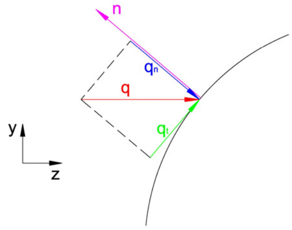


Figure 2: Heat flux vector decomposition.

Unlike the sides of the mask taper, which have a constant normal vector, on the corner surface the normal vector varies at each point of the surface. ANSYS does not have a build-in command to apply a coordinate dependent heat flux vector. The solution of this challenge is developing a code to provide the surface heat flux at each node of the element where the heat load is applied.

NORMAL VECTORS COMPUTATION

After meshing, ANSYS Workbench allows the user to obtain information about the mesh such as: element number and its nodes numbers, and coordinates with respect to the specified coordinate system [3]. In this case the coordinate system is user defined and its origin is situated in the middle of the beam footprint with respect to the x and y coordinates. Knowing the coordinates of two points one can compute a vector connecting the two points. In the same way, having the coordinates of three nodes of a surface element, one can compute two vectors which have a common point, each vector having the tail at the common

* Work at the Advanced Photon Source is supported by the U. S. Department of Energy, Office of Science, Office of Basic Energy Sciences, under Contract No. DE-AC02-06CH11357.
† capatina@anl.gov

point and the head at the adjacent point. By definition the vector product of two adjacent vectors is a vector which is perpendicular to the surface defined by the initial two vectors and having the tail on the common point. The new vector divided by its length gives us the normal vector to the surface in that particular point. Based on this theory, the normal vector at each node of the beam footprint is computed [4]. In order to loop through all the elements, we need to know how many elements there are in the selection. The following command is issued in order to obtain the number of elements:

*get,ne,elem,0,count (2)

Parameter ne is the user defined parameter name which stores the number of elements.

If we set the element number, e, to equal zero, the following command allows the user to get the next higher element number above e.

*get,e,elem,e,nxth (3)

The routine then loops through all the elements of the selection to compute the heat flux at each node.

The core of the routine starts with obtaining the node numbers and the x, y, z coordinates of each node of the element as shown by the commands (4) through (7). There are eight nodes per element for the chosen SURF152 elements, which are quadratic elements with middle side nodes, as shown in Fig.3.

*get,n1,elem,e,node,1 (4)
*get,n1x,node,n1,loc,x (5)
*get,n1y,node,n1,loc,y (6)
*get,n1z,node,n1,loc,z (7)

In the above commands, n1 is the node number of the first node of the element number e and n1x is the x coordinate of the node number n1, etc. After obtaining all the 8 times 3 coordinates for the first element, one can compute the normal vectors, make the projections and calculate the heat load at each node.

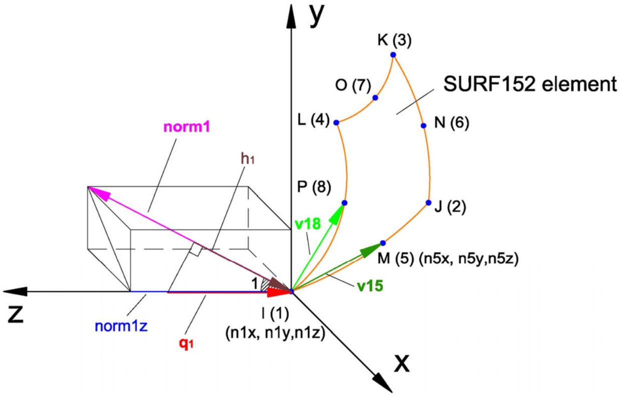


Figure 3: Normal vector at node 1 of a surface element.

The first normal vector to be computed, norm1, see Fig.3, has the tail on node 1 and it is perpendicular on v15 and v18. The vectors v15 and v18 are connecting node 1 with its adjacent nodes: 5 and 8, respectively. Vectors v15 and v18 have v15x, v15y, v15z, and v18x, v18y, v18z as coordinates, respectively. The coordinates are computed as follows:

$$v15x=n5x-n1x \tag{8}$$
$$v15y=n5y-n1y \tag{9}$$
$$v15z=n5z-n1z \tag{10}$$
$$v18x=n8x-n1x \tag{11}$$
$$v18y=n8y-n1y \tag{12}$$
$$v18z=n8z-n1z \tag{13}$$

The vector product definition is used to find the normal vector coordinates norm1 (norm1x, norm1y, norm1z):

$$v15 \text{ x } v18 = norm1 \tag{14}$$

In matrix notation this becomes:

$$\begin{bmatrix} e_1 & e_2 & e_3 \\ v15x & v15y & v15z \\ v18x & v18y & v18z \end{bmatrix} = \begin{bmatrix} norm1x \\ norm1y \\ norm1z \end{bmatrix} \tag{15}$$

where e1, e2 and e3 are the unit normal vectors of the coordinate system with the origin at node 1. Therefore, the normal vector, norm1, coordinates are:

$$norm1x = (v15y * v18z) - (v18y * v15z) \tag{16}$$
$$norm1y = (v18x * v15z) - (v15x * v18z) \tag{17}$$
$$norm1z = (v15x * v18y) - (v18x * v15y) \tag{18}$$

The modulus of the normal vector, norm1, is:

$$\text{mod\_}norm1 = \sqrt{norm1x^2 + norm1y^2 + norm1z^2} \tag{19}$$

The cosine of the angle between the normal vector, norm1, and the z direction can be now computed. This angle is called angle 1, as it refers to the normal vector at node number 1:

$$\cos 1 = \frac{|\,norm1z\,|}{\text{mod\_}norm1} \tag{20}$$

The heat load as provided by SRUFF at node 1, q_{n1}, is obtained by substituting x with n1x and y with n1y in equation (1). Finally, the length of the normal component of the heat flux, q_{n1}, at node 1, h1, is obtained by multiplying q_{n1} by cos1.

Following the same procedure, the other normal components at the nodes 2 through 8, hi, can be computed. Having all the hi values for the element e allows issuing the load command:

sfe,e,1,hflux,,h1,h2,h3,h4,h5,h6,h7,h8 (21)

This way each node of the selection is loaded with the appropriate heat flux value.

Content from this work may be used under the terms of the CC BY 3.0 licence (© 2018). Any distribution of this work must maintain attribution to the author(s), title of the work, publisher, and DOI.

ISBN: 978-3-95450-207-3 doi:10.18429/JACoW-MEDSI2018-WEPH07

Content from this work may be used under the terms of the CC BY 3.0 licence (© 2018). Any distribution of this work must maintain attribution to the author(s), title of the work, publisher, and DOI.

METHOD FOR ANSYS WORKBENCH

With appropriate pre-processing, the numerical algorithm described above can be used in ANSYS Workbench as a command snippet to apply the heat flux load.

Pre-processing

The above algorithm is written for quadratic elements and therefore the hex dominant method for meshing has to be used. To be able to capture the heat load accurately it is essential to fine mesh the surfaces of interest.

The user has to define a coordinate system, as shown in Fig. 4, with the origin in the middle of the beam footprint and assign a number, 12 for instance, as the APDL name.

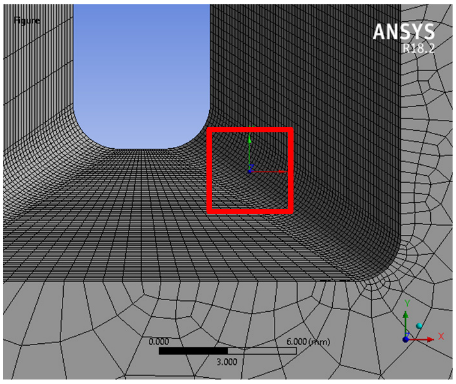


Figure 4: The mesh of the taper corner.

The surfaces of interest, in this case the corner and the adjacent sides, are selected and saved in a "Named selection" called “E_Surf”. By right-clicking on the named selection just created and choosing "Create Nodal Named Selection", the user creates a collection of nodes called “E_Surf_Nodes”.

In this step, the user also assigns the material, the heat transfer coefficient and the bulk temperature.

Command Snippet

The first step is to set the coordinate system to 12, then add SURF152 surface elements to the thermal analysis and enable heat flux input. Next, a subset of nodes corresponding to the beam size at the component or slightly larger is selected. The surface effect elements, SURF152, are created using ESURF and then selected as follows:

```
CSYS,12                                   (22)
*GET,ID_MAX_TYPE,ETYP,0,NUM,MAX           (23)
ET,ID_MAX_TYPE+1,152                      (24)
KEYOPT,ID_MAX_TYPE+1,8,1                  (25)
TYPE,ID_MAX_TYPE+1                        (26)
CMSEL,S,E_Surf_Nodes                      (27)
NSEL,R,LOC,X,-1.43,1.43                   (28)
NSEL,R,LOC,Y,-1.45,1.45                   (29)
ESURF                                     (30)
ESEL,S,TYPE,,ID_MAX_TYPE+1                (31)
```

Then the number of selected elements, ne, are obtained using equation (2), the element number, e, is set to zero and the coefficients a through g are defined. The core of the algorithm, consisting of a do – enddo loop through all of the selection elements, starts with equation (32). Following are equations (3) through (13) and (16) through (21), for the nodes 1 through 8. Equation (33) closes the loop.

```
*do,i,1,ne                                (32)
*enddo                                    (33)
```

Solution

The temperature distribution within the mask body is obtained by running the solution. The temperature plot for the example chosen here is shown in Fig. 5.

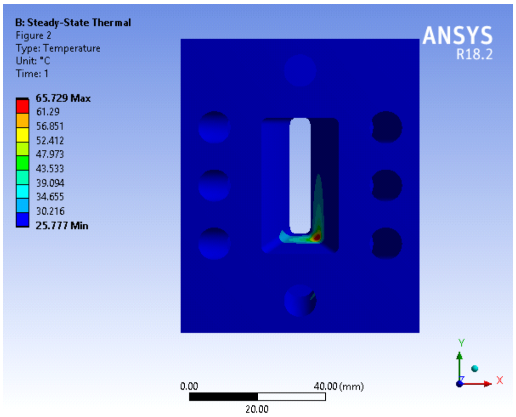


Figure 5: Temperature distribution.

CONCLUSION

The purpose of this work was to calculate the temperature distribution on a mask component subject to x-ray beam interception by means of using the commercial FEA software ANSYS Workbench of ANSYS, Inc. The analysis technique was described in detail. A sophisticated command snippet was developed. This allows applying the heat flux load generated by the x-ray beam on any surface type of a body. The technique described in this work is applicable to any component subject to x-ray beam interception. The prediction of the temperature distributions is very accurate; therefore, it is a reliable tool for the component validation procedure.

REFERENCES

[1] Y. Jaski *et al.*, “Thermomechanical analysis of high-heat-load components for the canted-undulator front end,” in *Proc. MEDSI2002*, Argonne, IL, USA, Sep. 2002, pp. 390-397.

[2] M. Meron, “SRUFF: A Comprehensive Package for Synchrotron Radiation Spectral and Optics Calculations,” unpublished.

[3] ANSYS Inc., “Introduction to ANSYS 9.0 – Part 1: training manual”, unpublished.

[4] D. Capatina, “A computational and experimental investigation of the heat flux applied by a photon beam,” M.S. thesis, Materials and Aerospace Engineering, Illinois Institute of Technology, Chicago, USA, 2008.